VOLUME II

THE PADS™-2000 REFERENCE MANUAL

CAD SOFTWARE, INC.
119 Russell Street
Suite #6
Littleton, MA 01460

(508)486-9521

Rev 1 10/01/90
Rev 0 06/15/90

TABLE OF CONTENTS

CHAPTER 1 - INTRODUCTION 1
1.1 Contents of Volume II, Reference Manual 1
1.2 Organization of the Software 4
1.3 PADS-2000 System Environment 5

CHAPTER 2 - GRAPHICS PRESENTATION AND OPERATION 7
2.1 Starting PADS-2000 7
2.2 Features of the Presentation 9
2.2.1 Cursor 10
2.2.2 Working Area 11
2.2.3 The Prompt Line 12
2.2.4 Cursor Location Indicator 12
2.2.5 Information Status Window 12
2.2.6 Selecting Commands from the Menu 15
2.3 Controlling the Graphics Window 16
2.3.1 Repainting the Viewing Window 17
2.3.2 Resetting the Viewing Window 17
2.3.3 Panning the Viewing Window 18
2.3.4 Zooming In 18
2.3.5 Zooming Out 19
2.3.6 Moving the Display Window 19
2.3.7 Defining the Viewing Window 19
2.3.8 Stopping the Display Repaint 19
2.4 Modeless Commands 20

CHAPTER 3 - THE IN/OUT MENU 25
3.1 Binary Job 25
3.2 IN/OUT 26
3.3 Loading a Design 27
3.4 Storing a Design 28
3.5 Reading an ASCII File 30
3.6 Creating a PADS ASCII File 31
3.7 Loading a FutureNet 35
3.8 Loading an Orcad Netlist 43
3.9 Converting a Redac Design 44
3.9.1 Meaning of Error Codes in Redac Conversion 45
3.10 Schema In 48
3.11 On-The-Fly 49
3.11.1 Adding a Connection 50

3.11.2 Deleting a Connection....51
3.11.3 Renaming a Net with On-the-Fly....52
3.11.4 Removing a Pin from a Net with On-the-Fly....53
3.11.5 Adding a Part....54
3.11.6 Deleting a Part with On-the-Fly....55
3.11.7 Renaming Parts with On-the-Fly....56
3.11.8 Modifying Tracks with On-the-Fly....57
3.11.9 Adding Tracks with On-the-Fly....57
3.12 Erase Design....58

CHAPTER 4 - THE SETUP MENU....59
4.1 The Display....60
4.1.1. Color Selection....60
4.1.2 ForGnd....61
4.1.3 Selected Color Indicator....61
4.1.4 Level....61
4.1.5 Displaying Parts....62
4.1.6 Connections....62
4.2 Defining Pad Stack Sizes....63
4.2.1 Adding Padstack Definitions....67
4.2.2 Deleting a Padstack....67
4.2.3 Adding a Level Definition....68
4.2.4 Deleting a Level Definition....68
4.2.5 Changing a Padstack....69
4.2.6 Completing the Padstack Definition....69
4.3 Changing the Size of Text....69
4.3.1 Changing the Width of the Board Outline....70
4.3.2 Changing the Width of the 2-D Lines....71
4.3.3 Changing the Line Widths in Part Decals....71
4.3.4 Changing Tracks Globally....71
4.3.5 Defining the Width of Non-Poured Copper Areas....71
4.3.6 Poured Copper....72
4.3.7 Setting the Size of Reference Designators....72
4.3.8 Setting the Size of Other Text....72
4.4 Setting System Parameters....72
4.4.1 The Show Level....73
4.4.2 Highlight....74
4.4.3 Displaying Tear Drops....74
4.4.4 Setting the Display Mode....75
4.4.5 Setting the Displayed Dot Grid....76
4.4.6 Setting Tracks and Lines to Real Width....76
4.4.7 Setting Thermal Line Width....76

4.4.8 Setting the Hatch Grid 77
4.4.9 Copper Pour Clearance 77
4.4.10 Design Rule Checking 77
4.4.11 Setting the Interval Between Backups 77
4.4.12 Setting the Maximum Layer 78
4.5 Setting Net Display 79
4.5.1 Selecting the Nets to Display 80
4.5.2 Controlling the Display of Nets 81
4.5.3 Selecting the Length Minimization 81
4.5.4 Setting Copper Share 83
4.5.5 Defining the Power Plane 83
4.5.6 Defining the Nets to be Routed by PADS-SuperRouter 83
4.5.7 Setting the Rip Up 83
4.5.8 Setting the Shove Rules 83
4.6 The Macro Command Language 84
4.6.1 Creating a Macro 84
4.6.2 Display Resolution Independence and Coordinates Within 85
4.6.3 Absolute Versus Relative Coordinates 85
4.6.4 The Auto Macro 86
4.6.5 Loading a Macro File 86
4.6.6 Macro Replay 86
4.6.7 Yes/No Questions 86
4.6.8 Yes/No Restore Mode 87
4.6.9 Macros Can Call Other Macros 88
4.6.10 Macro Commands 88
4.6.11 Create Auto/User Macro 88
4.6.12 Set Absolute/Relative Coordinate Mode 89
4.6.13 Restore Menu Position 89
4.6.14 Top Menu 89
4.6.15 Display File 89
4.6.16 Display Errors 90
4.6.17 Set Macro Replay Count 90
4.6.18 Help/Status 90
4.6.19 Load File 91
4.6.20 Save File 91
4.6.21 Delete a Macro 91
4.6.22 Defaults 91
4.6.23 Limits 92
4.6.24 Macro Language 92
4.6.25 Sample Macros 94
4.6.26 Modeless Commands 95
4.7 Setting the System Units 96

4.8 Establishing the X,Y Origin....97
4.9 Setting the Parameters for PADS-SuperRouter....97

CHAPTER 5 - THE CREATE MENU....101
5.1 Objective....101
5.2 The Text Command....102
5.2.1 Creating a New Text Item....102
5.2.2 Copying a Text Item....104
5.2.3 Modifying a Text Item....104
5.2.4 Deleting Text....104
5.3 Creating a Board Outline....105
5.3.1 Creating Non-Circular Board Outlines....105
5.3.2 Creating Circular Board Outlines....106
5.3.3 Loading an Existing Board....107
5.3.4 Commands for Modifying Board Outlines....107
5.3.4.1 Moving a Corner....108
5.3.4.2 Moving a Segment....108
5.3.4.3 Cutting Segments in a Line....109
5.3.4.4 Adding Additional Corners to a Board....110
5.3.4.5 Deleting a Corner in the Board Outline....110
5.3.4.6 Changing the Line Width for the Board Outline....111
5.3.4.7 Adding Arcs in a Board Outline....111
5.3.5 Moving the Board Outline....111
5.3.6 Deleting a Board Outline....112
5.3.7 Saving the Board in the Library....112
5.4 Copper Areas....113
5.4.1 Creating a Copper Area....114
5.4.2 Copying a Copper Area....116
5.4.3 Modifying a Copper Area....116
5.4.4 Moving a Copper Area....117
5.4.5 Deleting a Copper Area....117
5.5 Flooding a Poured Copper Area....117
5.6 Creating Cut Outs in Poured Copper....118
5.7 2-D Lines and Shapes....119
5.7.1 Creating a 2-D Line....120
5.7.2 Creating a Circular 2-D Item....120
5.7.3 Adding 2-D items from the Library....121
5.7.4 Copying 2-D Items....121
5.7.5 Modifying 2-D Items....121
5.7.6 Moving 2-D Items....122
5.7.7 Deleting 2-D Items....122
5.7.8 Combining 2-D Lines and Text....123

5.7.9 Saving a Drawing Item in the Parts Library 123
5.8 Creating Part Decals 124
5.8.1 Creating a New Part 125
5.8.2 Creating the Decal Outline 126
5.8.3 Copper in Decals 127
5.8.4 Adding Text to the Decal 128
5.8.5 Creating a Copper Cutout 129
5.8.6 Moving the Default Part Name Location 129
5.8.7 Placing Terminals Pads 129
5.8.8 The Setup Menu 130
5.8.8.1 Controlling the Display 131
5.8.8.2 Changing the Origin of the Decal 131
5.8.8.3 Changing the Display Dot Grid 132
5.8.8.4 Max Layer (Multilayer Part Decal 132
5.8.8.5 Setting the Units 132
5.8.9 Saving the Decal 133
5.8.10 Modify an Existing Part Decal 134
5.9 Part Types 134

CHAPTER 6 - PLACING PARTS 135
6.1 Move vs.Matrix Move 136
6.1.1 Rotating a component while in the Move menu 138
6.1.2 Placing a Component on the Opposite Layer 139
6.1.3 Displaying the Minimum Connection Length 139
6.1.4 Working with Alternate Parts 140
6.1.5 The Matrix Command 140
6.1.6 Abort or Exit (While in Move) 141
6.1.7 Completing the Placement 141
6.2 Rotating Components 141
6.3 Placing Components on the Opposite Layer 142
6.4 Fixing and Unfixing Components 142
6.5 Moving Component Names 143
6.6 Autoplacement 144
6.7 Creating Placement Matrices 144
6.8 Placing Components on a Matrix 146
6.9 Optimizing the Placement 149
6.9.1 Swap Parts 149
6.9.2 Swapping Gates 151
6.9.3 Swapping Pins 152
6.10 Autoplacement of Discrete Parts 154
6.11 Establishing the Minimum Connection Pattern 155
6.11.1 Determining the Minimum Net Length 155

6.11.2 Setting the Length Minimization Rules 156
6.11.3 Biasing Connections for Autoplacement 156
6.12 Evaluating the Placement Quality 157
6.12.1 Calculating the Global Net Length 157
6.12.2. The Connection Density Histogram.....157
6.12.3 The Connection Density Map.....158
6.13 Component Placement by Group 159
6.13.1 Defining a Group.....159
6.13.2 Working with Groups 160
6.13.3 Using Groups in Other Designs.....161
6.14 Auto Rename of Components.....162

CHAPTER 7 - TRACK ROUTING.....165
7.1 Interactive Routing.....166
7.2 Converting a Connection to a Track.....167
7.2.1 Adding Corners While Routing.....169
7.2.2 Deleting a Corner While Routing 169
7.2.3 Making Any Angle Corner in Routes.....169
7.2.4 Changing Layers in a Route 170
7.2.5 Ending the Track.....170
7.2.6 Finishing the Route (FIN PIN 171
7.2.7 Aborting a Track Route 171
7.2.8 T-Routing.....172
7.2.8.1 T-Routing Features.....173
7.3 Modifying Routes 173
7.3.1 Move Corner 174
7.3.2 Move Segments in a Track.....174
7.3.3 Cut Segment Option 174
7.3.4 Adding a Corner.....175
7.3.5 Deleting a Corner 175
7.3.6 Changing the Via Type.....176
7.3.7 Changing the Track Line Width.....176
7.3.8 Changing the Level of a Segment 177
7.3.9 Converting Routes Back to Connections.....178
7.4 Autorouting with PADS-2000 178
7.5.1 Selecting Grid, Layer Pair and Via Type for Routing.....179
7.5.2 Selecting Visible Nets 179
7.5.3 Selecting the Routing Grid.....180
7.5.4 Selecting Individual Router Passes 180
7.5.5 Selecting Connections to be Routed 183
7.5.6 Learning to Use the Autorouter 184
7.6. The Power and Ground Router.....187

7.6.1 Theory of Operation 187
7.6.2 Options Supported 188
7.6.3 Power-and Ground Router Limitations 188
7.6.4 Power and Ground Router Check List 189
7.7 The Memory Router 191
7.7.1 Theory of Operation 191
7.7.2 Options Supported 191
7.7.3 Memory Router Limitations 192
7.7.4 Memory Router Check List 192
7.8 The Maze Router 193
7.8.1 Theory of Operation 194
7.8.2 Options Supported 194
7.8.3 Maze Router Limitations 196
7.8.4 Maze Router Check List 196
7.9 Determining the Length of a Track 198
7.10 The Shove Router 199
7.10.1 The Shove Menu Options 200
7.10.2 The Auto Path Router 200
7.10.3 The Auto Steer Router 201
7.10.4 Routing with On-Line DRC 202
7.10.5 Checking with the Shove Router 203
7.10.6 The Strategy File 203
7.10.6.1 STRAT IN 203
7.10.6.2 STRAT OUT 204
7.10.6.3 MOD RULES 205
7.10.6.4 The Standard Strategy File 208
7.10.6.5 MOD NETS 209
7.10.7 Other Commands in the Shove Router 211
7.11 Selecting Tear Drop Pads 211

CHAPTER 8 - CHECKING THE DESIGN 213
8.1 Objective 213
8.2 Design Rule Check 213
8.2.1 Suggested Usage 215
8.3 Check ASCII 215
8.4 Check Future Net 216
8.5 Check Schema 216
8.6 Check Plane 217

CHAPTER 9 - THE ENGINEERING CHANGES (ECO) MENU 219
9.1 Making an ECO in the Board 220
9.1.1 Changing a Part Type 220

9.1.2 Adding a Route....222
9.2. Updating a Design with an ECO from the Schematic....222

CHAPTER 10 - PRODUCING REPORTS....225
10.1 Objective....225
10.2 Reports Menu....225
10.3 Net List Report....226
10.4 Unused Report....226
10.5 Statistics....227
10.6 JOB LIMITS....228
10.7 PART LISTS....231

CHAPTER 11 - CAM OUTPUTS....233
11.1 The CAM Outputs....233
11.2.1 Matrix Printer Plots....233
11.2.2 Wet Ink Plots....233
11.2.3 Photoplotter....234
11.2.4 Laser Printer....234
11.2.4.1 Postscript Laser Printers....234
11.2.5 Drill....235
11.3 Using the CAM Menu....235
11.4. Outputting Directly to Plotters....236
11.5 Batching CAM Outputs....250
11.6 Changing CAM Defaults....251
11.6.1 Defaults - Misc....251
11.6.2 Defining the Default Photoplot Aperture Settings....254
11.6.3 The N/C Drill Defaults....255
11.7 Verify....257
11.8 CAM Output Files....257
11.9 Wet Ink Quality Plots....259
11.10 The N/C Drill Output....260
11.11 The DRILL.DAT File....266

CHAPTER 12 - SYSTEM PARTS LIBRARY....269
12.1 2-D LINES Database....271
12.2 PART DECALS Database....271
12.3 PART TYPE Database....271
12.4 PADS-LOGIC DECAL Database....272
12.5 Use of the PLIB EDITOR....272
12.5.1 P (Part Type Library)....273
12.5.2 Part Type (PLIB Format)....277
12.6 PART TYPE (CREATE, PART TYPE Format)....287

12.6.1 NEW PART 288
12.6.2 GATE INFO 289
12.6.3 NEW CONN 290
12.7 Initialize 291
12.8 System Parts Library 292
12.9 Maintenance of Systems Parts Library 292
12.9.1 Creating or Modifying Part Decal/Types 292
12.9.2 Making Backups of the Library 293
12.10 Discrete Part Naming Convention 294

CHAPTER 13 - ASCII FILES FORMAT 297
13.1 Objective 297
13.2 The ASCII In File 297
13.3 Manually Creating a Board Database Input ASCII File 298
13.3.1 Board Outline 298
13.3.2 Parts List 299
13.3.2.1 Notes on Parts List Files 301
13.3.2.2 Example Part List 302
13.3.2.3 Parts List Error Message 304
13.3.3 Connection List 304
13.3.3.1 Notes on Connection List 305
13.3.3.2 Example of a Connection List 309
13.3.4 Connection List Error Messages 309
13.4 The ASCII OUT File 310
13.5 The STARTUPL.ASC File 311
13.6 The Editor (PEDIT) 312

CHAPTER 14 - FILES MANAGEMENT 315
14.1 Objective 315
14.2 Job Directory Listings 315
14.3 Job/Disk Management and Control Procedure 316
14.4 Deletion of Files 317
14.5 Daily Backup 318
14.5.1 Making a Backup Floppy 318
14.5.2 Using the Backup Disk 319
14.6 Archival Files 320
14.7 Storage Space 321
14.8 Software Updates 321

APPENDIX A1 PADS-SUPERROUTER
APPENDIX A2 Background Information for Photoplotting
APPENDIX A3 ECO Forward/Backward Annotation Format

APPENDIX A4 A Summary of athe PADS Library Concept
APPENDIX A5 Networking PADS-2000
APPENDIX A6 System Files
APPENDIX A7 Error Messages
APPENDIX A8 PADS-PCIF
APPENDIX A9 Parts List and Decal Plots
APPENDIX A10 PADS-THERMAL
APPENDIX A11 PADS-HIRES
APPENDIX A12 PADS-DXF

CHAPTER 1 - INTRODUCTION

This is the second of the two-volume PADS-2000 User Manual. These two volumes have been developed with the goal of providing a self-training course and a ready reference for the user.

VOLUME I, The PADS-2000 SELF-TRAINING MANUAL, contains a series of tutorials that rapidly acquaint the user with the system by "actual use" of the system in practice sessions.

VOLUME II, the PADS-2000 REFERENCE MANUAL, has the objective of providing extensive detailed information on a subject, when the user feels the need to refer to more detailed information than as contained in the Self-Training Manual.

1.1 Contents of Volume II, Reference Manual

This introductory chapter describes the contents of each of the chapters and the appendix of the Reference Manual.

In general, the chapters are arranged in the sequence in which you will design a board, i.e., learning the graphics presentation will be first, followed by each of the major menu commands in the sequence they would normally be used in designing the board. Use this introductory chapter to become familiar with where to find a subject in the manual.

Chapter 2 - Graphics Presentation

This chapter describes in complete detail the graphics presentation, the significance of the various pop-up windows, and how you manipulate the display in terms of enlarging or contracting the view, moving the view of the working area.

Chapter 3 - The Input/Output Menu

This chapter discusses the options of the Input/Output menu. This menu is used to load and store designs on the disk, to produce complete ASCII data files of the design

Chapter 4 - The SetUp Menu

The commands in this menu enable the user to define the various parameters for the design such as the display color of the different items the number of routing layers of the board, the spacing of the CRT dot grid, size of draw lines and text, DRC spacing rules, dimension unit, and other system parameters.

Chapter 5 - The Create Menu

Chapter 5 discusses the commands associated with the Create Menu. The Create Menu contains commands for the creation of 2-D lines, Board Outlines, Copper areas, Part Decals and General Text.

Chapter 6 - Placing Parts

The commands of the Placement Menu are used by the designer to arrive at an optimum placement for the components on the board. Components may be placed on both front and back layers as would be the case with Surface Mounted Devices. Both Interactive and Automatic Placement commands are available, as well as placement quality measurement tools.

Chapter 7 - Track Routing

This chapter discusses the commands in PADS-2000 used for the Interactive and Automatic routing of the connections on the board.

Chapter 8 - Checking the Design

After the routing of the connections has been completed, it is necessary to insure the air gap or spacing requirements between the various pads, tracks, vias, etc., have not been violated. The Check Menu commands are used for this function. Check also contains a comparison check that permits the checking of the connectivity current database against the original input data.

Chapter 9 - The Engineering Changes (ECO) Menu

This chapter discusses the commands used for modifying an existing design database. Connections, Components, and Part Types can be added, deleted or changed.

Chapter 10 - Producing Reports

Various hardcopy reports can be obtained including Part List, Netlist, job statistics, etc.

Chapter 11 - CAM Outputs

This menu is used to generate the Computer-Aided- Manufacturing tools including artwork, drawings, drill drawings, silkscreen, pads only, artwork and drill tapes.

Chapter 12 - The System Parts Library

This chapter discusses the use of the System Parts Library. The System Library contains an editor that allows the designer, using this editor, to create these parts, edit, modify, and store them for subsequent use.

Chapter 13 - ASCII Format

PADS-2000 has a specific ASCII data format that may be used to create an ASCII design database for transfer to other CAD environments. This chapter describes the use of the PADS-2000 program database ASCII.

Chapter 14 - Files Management

This chapter discusses the control and storage of the files created with the system.

Appendices
Appendix A1 - PADS-SUPERROUTER
Appendix A2 - Background Information for Photoplotting
Appendix A3 - ECO Forward/Backward Annotation Format
Appendix A4 - A Summary of the PADS Library Concept
Appendix A5 - Networking PADS-2000
Appendix A6 - System Files
Appendix A7 - Error Messages

Appendix A8 - PADS-PCIF
Appendix A9 - Parts List and Decal Plots
Appendix A10 - PADS-THERMAL
Appendix A11 - PADS-HIRES
Appendix A12 - PADS-DXF

1.2 Organization of the Software

The software will be loaded onto your personal computer with a special installation program. All the programs and data files associated with PADS-2000 are contained within a single Directory Path .

Figure 1.1 illustrates the programs and subdirectories associated with the PADS-2000.

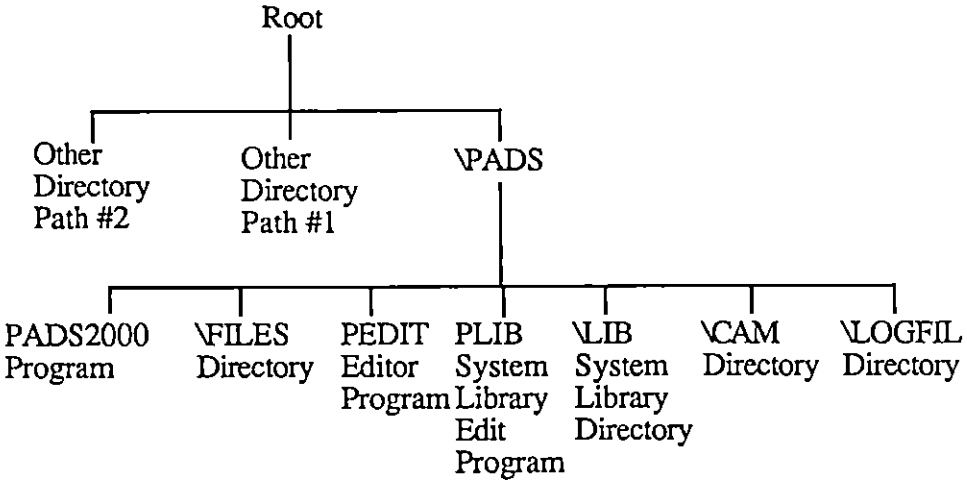


Figure 1.1 - DIRECTORY PATH

The PADS directory resides on the hard disk with other directories. It should be located in the root directory.

All of the PADS design programs are stored in the \PADS directory. Under the PADS directory, there are associated sub-directories. The directories are (1) the \FILES directory for storing jobs produced in the

PCB design program, (2) the \LOGFIL directory for storing data produced by the optional PADS-Logic program (3) the \CAM directory for the output files of the CAM commands.

1.3 PADS-2000 System Environment

PADS-2000 is a virtual memory product running under MS-DOS. This is in some ways a contradiction, because MS-DOS is not a virtual memory operating system. This has been achieved by using special protected mode system commands in PADS-2000. This may make PADS-2000 incompatible with other non-PADS software resident in your system.

You should check that your system is configured in conformance with the following guide lines:

1) PADS-2000 assumes it has access to the entire memory area in your system. No TSR (Terminate and Stay Resident) programs like Sidekick may run concurrently with PADS-2000.

2) PADS-2000 is not compatible with EMS expanded memory system driver packages like 386MAX or QEMM. If you are using these packages, you must be sure to turn them OFF before entering PADS-2000. Most EMS drivers can be turned on and off. This may be done by making modifications in the PADS.BAT file that is invoked when you enter PADS programs.

3) PADS-2000 may not be compatible with your systems networking software.

4) PADS-2000 will use disk space for database swapping if the memory space you require is not available in physical memory. This disk space is used as Virtual memory. Using virtual disk memory leads to a decrease in the software speed performance, so if you intend to do boards that exceed the physical memory of your system, we recommend that you purchase additional extended memory.

You will be asked to define the amount of memory that is to be allocated to PADS-2000, with the prompt:

How many MBytes of memory to allocate? [1-20]:

Respond with a number in the range of 1 to 20, followed by <cr>. This will be the available memory for your database. If you choose a memory allocation size that is insufficient for the design you wish to create, an Out of Memory message will appear and you will be required to exit PADS-2000 to increase the database size. In addition, the PADS-2000 program and swap files will require a further 1512 KB of memory.

The copyright screen will then appear, together with the software version number.

A message is output that defines the memory requirements, based on the size of the program and the size of the database area you require. If the memory size you choose exceeds the physical memory available, PADS-2000 will automatically define a portion of the hard disk for the database and will do its own disk swapping. This will allow a larger board to be designed, but with a penalty of slower operation. The amount of disk space being used is listed as part of the message.

Total Memory Available N

System Memory Used N
Program Memory Used N
Database Memory Used N
Display List Memory Used N
Total memory Used

RAM Memory Used N
Disk Memory Used N

Equivalent to board with N parts and M nets with P connections.

Press any key to continue

Press any key, and PADS-2000 is loaded into memory. The CRT presentation shown in Figure 2.1 will appear after a few seconds.

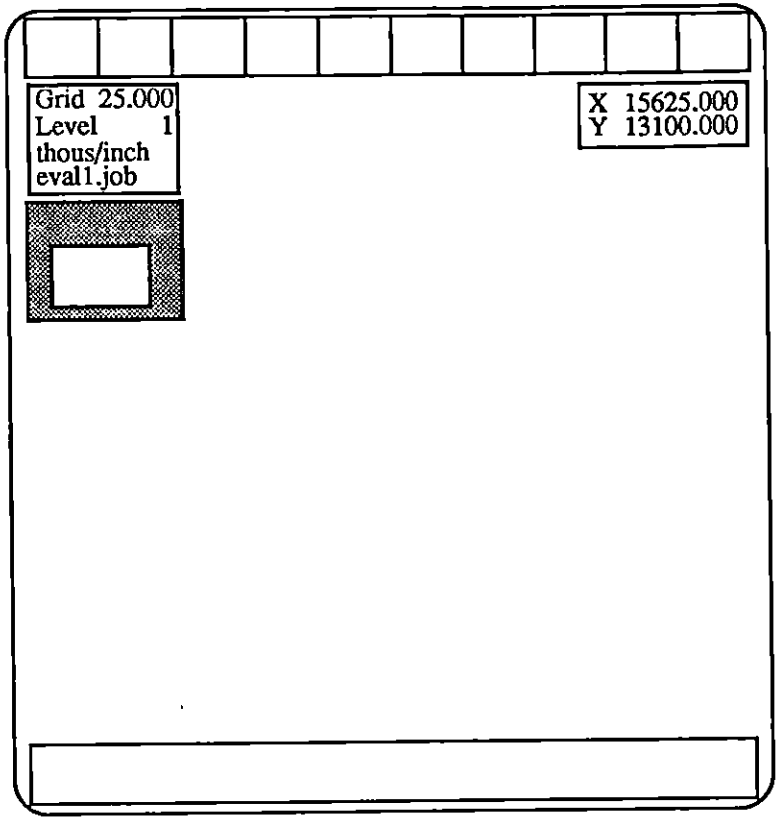


Figure 2-1 INITIAL CRT PRESENTATION

2.2 Features of the Presentation

Figure 2.2 and the following discussion, identifies and describes the different items presented on the CRTTerminal.

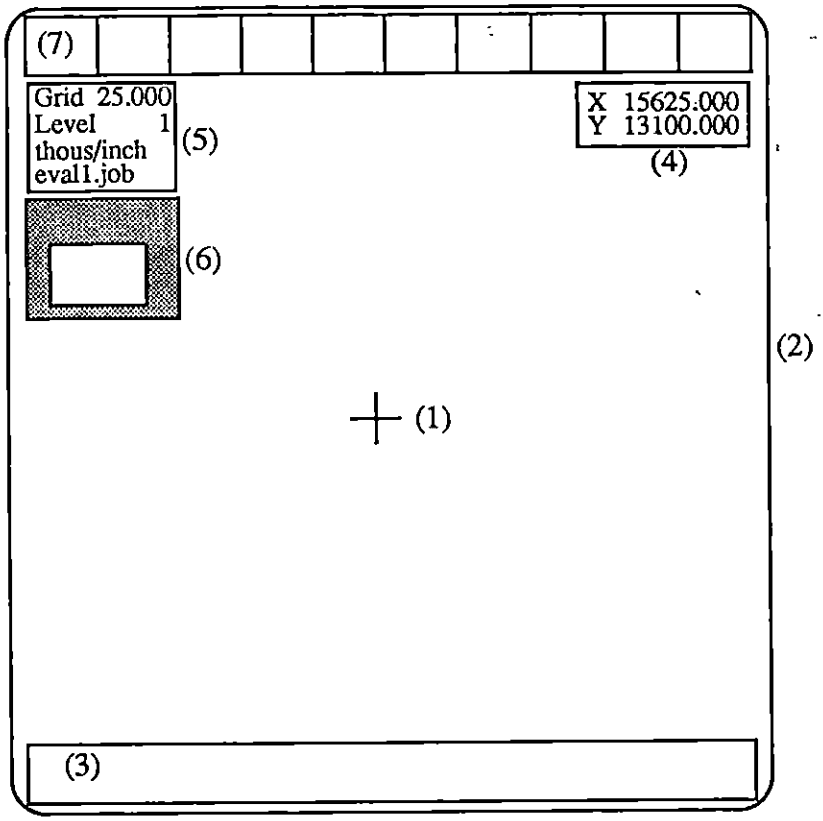


Figure 2.2 - CRT FEATURES

2.2.1 Cursor

(Item 1 in Figure 2.2)

The cursor consists of a set of crosshairs or an arrow that is highlighted. The cursor movement is generated by moving the mouse.

2.2.2 Working Area

(Item 2 in Figure 2.2)

The working area is the portion of the CRT where the board is designed; it is your Drafting Table.

The system database and Working Area has a maximum X and Y value of 56.3" or 143 cm. The database is a square area. The database has a resolution of 0.00001" or .001 mm. Note that the actual system grid, as stored within the database, is actually smaller than 10 to the -7 cm. We limit the user to inputting data with an accuracy of .00001" or .0001mm in order to guarantee exact translation of coordinate values when the user switches between metric and inch coordinates.

In PCB design, one designs the board either by looking down on the components (Component Side Convention) or by looking down on the solder side (Solder Side Convention). Although the PADS-2000 can use either convention, it is strongly advised that you use the Component side convention. All of the parts in the PADS Parts Library, have been created using the "Component Side" convention, and should not be used using the Solder Side convention.

In this convention, the designer would be looking down on the top of the components at the CRT; Layer one of the board would be the first level at the CRT (the component layer), and higher numbered layers would be in the dimension towards the inside of the CRT and going away from Layer one. The solder layer is generally the last layer that can have tracks, and this is adjustable for each specific design using the Maximum Layer command described in Chapter 4.

You will notice a dot grid displayed on the working area. The default spacing of the dots is 1 inch. This working grid is a very useful grid during the construction of lines and placing of terminal pads. The dot spacing can be changed, as appropriate by the designer.

2.2.3 The Prompt Line

(Item 3 of Figure 2.2)

The prompt line is the means of text communications between PADS-2000 and you. Request prompts for confirmations such as, "ARE YOU SURE YOU WISH TO DELETE COMPONENT XYZ, Y/N?," Potential Error Messages, etc., are displayed in this window. When you type information, it will appear on the prompt line as well, and will be entered when you select carriage return.

2.2.4 Cursor Location Indicator

(Item 4 in Figure 2.2)

It is possible to display the current location of the cursor at any time you wish in the upper right corner of the screen. This is done by typing:

CTRL-L

(Hold down the control key while pressing L). The current X and Y locations in the selected grid unit is displayed. The command is a toggle, so the the display may be turned off by reselecting CTRL-L.

Additionally, the display may be invoked by typing:

CTRL-PgDn

The current X and Y cursor location is displayed, along with the X and Y location relative to the last position selection. Selecting CTRL-PgDn again will toggle the relative X and Y location displays on and off.

2.2.5 Information Status Window

(Item 5 in Figure 2.2)

The Information Window can be displayed in the upper left corner of the screen at any time by typing:

CTRL-I

The command is a three way toggle. The first time CTRL-I is selected, the current menu path and values are displayed. The second time CTRL-I is selected, the "postage stamp" is displayed. The third time CTRL-I is selected, both displays are removed. Alternatively, the user may select the TAB key.

When an electrical item in the database is selected, information of the item will be displayed in this window. For example, picking a connection or track will display information, about the connection such as its end points, signal name, and track width. Selecting a part will display the part name (Ref Designation), part type, and technology. Picking a pin of a part will display the Part Name, Pin Number and Part Type. You will find this window extremely useful while designing the circuit.

This pop-up window displays a variety of information:

1) The name of the current menu path.The Command Menu Path identifies the currently selected menu and the path in the menu hierarchy used to get to the menu. The window changes with changes of the Command Menu.

2) The current value for four different items associated with the database: Grid, Level, System Units, and Job Name. At various times, the System units display is replaced with the current Width, and the Job Name is replaced with the Layer Pair and Via Type information. The meaning of the different parameters is discussed below.

Grid
This item indicates the currently selected system grid, which defines the minimum movement of the cursor. Any value between .00001" and 2.000" is valid if the current System Grid is inches. If the current system unit is millimeters, the accepted values are .001 to 50. The Grid is changed with the G modeless command. See section 2.4 for details.

Level
The Level display defines the current layer on which data will be entered. It is not always visible. PADS-2000 supports databases of 30 different levels or layers . The current layer is given in this display. The Level is changed with the L modeless command. See section 2.4 for details.

A Level can be used either as a signal layer, a "pads only" layer, a Silkscreen layer, a Drilling drawing, an Assembly drawing, a Ground or Power plane, etc., as determined by the designer. The current layer number indicated at the CRT can be considered as the active level. When drawing a line, entering text, etc., it is entered in the database on the active level. For example, if one wished to put text on Level 1, insure that current layer is set to 1 and enter the text. It will be placed only on Level 1. Levels 1 through 30 are allowed individual levels, or layers.

When creating either a drawing or text item, insure that the active level is set correctly before entering the data.

NOTE: you can change the layer after entering the item. This is explained in later chapters.

If you wish to place an item on all layers, you may do so by first setting the active layer to 0. Level 0 is an "ALL LEVELS" special case used for those situations in which data is to be placed on all levels. For example, the Board Outline normally is shown on all layers of artwork, as well as all drawing layers. When creating the Board Outline, the Default level is set automatically to Level 0. As a result, the Board Outline will then be entered on Level 1 through Level 30. For other items such as Text, Draw Lines, etc., that are desired on ALL levels, the Designer sets the Default Level to 0; and the entry will be placed on ALL levels.

NOTE: Copper may not be place on Level 0, only levels 1 through 30.

Width
The WIDTH parameter indicates the width, in either inches or millimeters, of the line currently being drawn, and is active and displayed when you are creating the board outline, a drawing item, a part decal outline, a copper area, routing a track, or adding a connection to the database. Setting Width values can be achieved by using the W modeless command. See section 2.4 for details.

Text
The Text data displayed for Text is used to set the Size (Height) and Line Width of all text items used during the design of the board. Text has two values, H for Height, and W for the line width used to draw

the text. The text values are changed with the T modeless command. See section 2.4 for details.

From time to time, you will ask PADS-2000 to perform a task that will take some time to complete. When PADS-2000 is processing data and will be busy for more than 10 seconds, a message "WORKING..." is displayed in the Information Window.

Postage Stamp
(Item 6 in Figure 7.2)

The Postage Stamp feature shows the current location of the working area relative to the overall PCB board outline. A filled rectangle representing that area displayed in the working area, and an empty rectangle displays the overall dimensions of the design.

Postage Stamp is a feature added to relate the presentation of board data at the CRT to the board outline. For example, a small part of the board has been expanded and shown on the CRT; it may not be possible to know where, on the board, the CRT presentation is located.

During interactive routing and component placement, the postage stamp will be replaced by information on the current track being edited or component being moved. The postage stamp will reappear when the command is completed.

2.2.6 Selecting Commands from the Menu

(Item 7 in Figure 2.2)

The Command Menu Line displays the current active menu, and is used to select the commands to control the system during the design activities.

The PADS-2000 Program uses a Hierarchical Command menu structure. The highest rank in the Hierarchy is the menu of Main Menus, which are nine different menus, plus EXIT, as follows:

1 In/Out	**2 Setup**	**2 Create**	**4 Place**	**5 Route**
6 Check	**7 ECO**	**8 Reports**	**9 CAM**	**10 Exit**

When any one of the nine Main Menus is selected, the Major Options menu is replaced with the menu corresponding to the selected command. The command menus in turn have from one to nine sub-menus. When a sub-menu option is selected, its command options will appear in the Menu window and may be selected.

The Command Menu items are selected as follows:

- The commands may be selected with the Function Keys, F1 through F10 on the keyboard. Before each option is a number, 1 through 10, which indicates the function key to use.

- Moving the cursor over an entry in the menu will highlight the command in red. Pressing the left mouse button will then select the command which is highlighted.

- For three button mice, the middle button #2 is used to move the Highlight Bar over the menu items. When the middle button is clicked, the Menu Highlight Bar advances to the next menu box. When it is depressed and held down, you will see that moving the mouse will move the Menu Highlight Bar back and forth on the menu.

A position with the left hand over the Function keys and the right hand on the mouse is the most comfortable and efficient operational position. With practice, the simultaneous selection of the menu commands with one hand while manipulating the mouse to select data in the working area will be the best way to use PADS-2000. This two-handed operation provides very efficient operation. Alternately, the designer can use only one hand and the mouse to position the cursor and to select menu commands. It is suggested that you attempt both methods, and utilize the technique best suited to you.

F10 or EXIT is the final option in all menus, and is used to exit that menu and go to the next higher menu. When F10 or EXIT is selected from the Major Option Menu, you will exit PADS-2000 and return to the DOS Operating System.

2.3 Controlling the Graphics Window

Located on the right-hand side of all standard personal computer keyboards is a Numeric Keypad containing Numeric keys 1 through 9.

To use the window commands, insure that the Num Lock light is Off. Eight of these Numeric Keys, numbers -- 1, 2, 3, 4, 6, 7, 8 and 9 are used for manipulating the CRT, as shown in the following Figure 2.3.

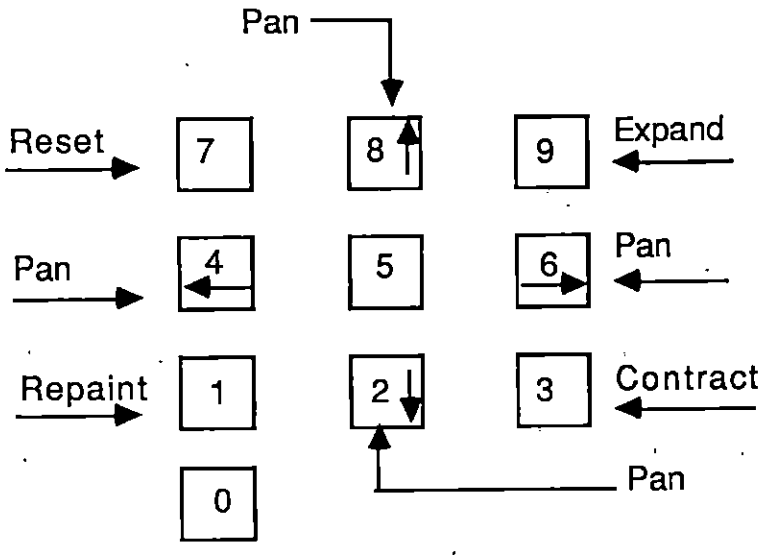


Figure 2.3 - CRT MANIPULATION KEYS

2.3.1 Repainting the Viewing Window

Keypad #1 is used to repaint the CRT screen without changing the viewing window. Repaint is used to fill in blanks in the display raster that are removed after components/connections have been deleted or moved. Repaint is also used after changing text size to present text at the new size.

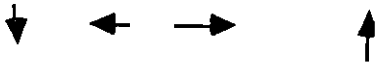
2.3.2 Resetting the Viewing Window

Keypad #7 is used to reset the window display. There are two Reset modes, depending upon whether a board outline is in the file being displayed. If there is no board outline present, pressing Numeric key

#7 will reset the presentation to the center of the CRT at the minimum scale.

If a Board Outline is present, pressing Numeric key #7 will Reset the display with the entire board displayed within the limits of the working area. The Reset key is particularly useful if you become confused as to where you are on the board. Selecting Numeric key #7 will Reset the viewing window display.

2.3.3 Panning the Viewing Window



These keys are used to move, or pan the viewing area around to different locations of the layout. In the process, it does not enlarge or contract the scale. To PAN, press Numeric keys 2, 4, 6 or 8, depending upon the direction you wish to pan. The zooming scale will remain unchanged.

When a pan key is selected, the viewing window moves in the direction on the arrow one half the width of the viewing window. This means in effect that the data located at the edge of the viewing area will appear in the center when the display is redrawn after a pan. The pan command has many uses: when you reach the edge of the viewing window while interactively routing a track, selecting a PAN key will move the track that is at the edge of the window to the center, so you can continue the Interactive routing process.

2.3.4 Zooming In

This key is used to "zoom in", or double the scaling factor of the data being displayed in the viewing area.

To use Zoom In, place the cursor at the area to be expanded. Press Numeric key button #9, and the item at the cursor location will be moved to the center of the CRT; and the data presented will be doubled in scale.

2.3.5 Zooming Out

This key is used to "zoom out" or reduce the data at the CRT by one half, centered on the current location of the cursor. To use Contract, place the cursor at the area to be contracted. Press Numeric key button #3, and the scale will be reduced in size by one-half (more data presented at the CRT).

2.3.6 Moving the Display Window

In addition to the four pan commands, you may also move the viewing area with this key. To use, move the cursor over the data which you wish to appear in the center of the graphic screen and press #0. The display is redrawn so that the area under the cursor is moved to the center of the viewing area.

2.3.7 Defining the Viewing Window

This function allows the designer to define a box area that will be magnified to fill the entire viewing window.

Select the DEL key. Place the cursor where you wish a corner of the window box to be created. Press mouse button #1 or F1 to START. The sides of the Window Box will start, and the menu option will change to COMPLETE. Drag the cursor to the right, either up or down, depending upon the direction of the window box you wish to create and press mouse button #1 to complete the window when the box has enclosed the desired area. The current display will be replaced with the contents of the Window Box enlarged to fill the entire viewing area.

2.3.8 Stopping the Display Repaint

It is possible to stop a repaint while it is occurring by selecting the ESC key.

NOTE 1: In some of the menus, the Window, Expand and Contract functions are also callable from Function Keys, in addition to the Numeric Keyboard comments described above.

NOTE 2: For very large boards, if a Window command is given that would move the board outside of the 40" x 40" database structure, then the result is modified to retain the entire board within the database structure. The result is that the window display that is being attempted will be modified from that described in the preceding.

NOTE 3: The Numbers Lock should be OFF for the Numeric Keys to function, as described above.

NOTE 4: The Window commands, are repeatable keys. That is to say, if you hold them down, they will accumulate a number of hits or commands, which will continue to be activated after you have released the key. Accordingly, it is good practice to learn to minimize the time the key is depressed, so that only one CRT manipulation occurs per key strike.

NOTE 5: Measure - CNTRL PGDN
When CNTRL & PGDN keys are selected together, Measure is switched on. This allows one to measure (X & Y grid window) the distance from the cursor position at the time of the selection, to the cursor position, as it is moved. Another CNTRL PGDN toggles the feature off.

2.4 Modeless Commands

The majority of the commands used to control the system are selected from PADS-2000's hierarchical menu structure. The menu hierarchy structure contains major menus, submenus, and a lower level of commands.

For some functions, there exists a method of bypassing the command menus, through the use of "Modeless Commands". Modeless Commands may be selected at any time by typing at the keyboard, without selection of a command using the menu.

You will find Modeless Commands of great benefit in speeding up your design effort. To assist you, the following is a summary of the Modeless Commands in the system:

1. The SEARCH command - used to search for an X-Y location in the database, for locating the cursor, precision placement of a Part, Drawing a 2-D Line, etc.

Use:

S XXX YYY <CR>

where
S is for Search
XXX is the X coordinate
YYY is the Y coordinate.

Use

S XXX <CR>

where
S is for Search
XXX is the reference designator name of the part to be located

Use

S XXX.P <CR>

where
S is for Search
XXX is the reference designator name
P is the pin number pin number of the part to be located.

NOTE: The Search command will only search for Pin Numbers and NOT Pin Names.

2. The LEVEL command- used to change levels or layers.

Use:

L X <CR>

where
L is for Level
X is the new Level.

3. The GRID command- used to change the system snap grid.

Use:

9. HIGHLIGHT A NET - used to highlight a Designated Net.

Use:

NXXX

where

N = NET Highlight switch
XXX = Name of net to be highlighted

An N without a Signal NET Name will turn OFF the current highlighted nets.

10. The REAL WIDTH command- used to define the maximum track width to be displayed with a center line only.

Use:

RXX

where

R = Real Width
XX = the track width in thousands of an inch above, which the track will be shown at the true width.

11. The BOXED TEXT command- used to turn on or off the box that can surround all text items and define the plotted limits of text entry.

Use:

X

to switch the current status of the command.

12. The FRACTIONAL GRID command- used to specify a grid as a fractional grid for the system.

Use:

GP N

where

GP= Fractional Grid
N= the fraction of the current unit to be selected. Allowed values are: 10, 12.5, 16.67, 20, 25, 50, 60, 80, 100, 200, 250, 400, 500. The grid is equal to the inverse of the current grid, e.g. 40 is 1/40" or .025", 1/60 is 1/60", or .016667".

CHAPTER 3 - THE IN/OUT MENU

The objective of this chapter is to familiarize the reader with the major functions of the In/Out menu of PADS-2000.

The IN/OUT (I/O) menu is at the top of the menu hierarchy. Its uses are as follows:

- to save a design that is in memory to the hard disk in a binary format.
- to load designs in binary format from the hard disk to memory and display them.
- to convert designs held in memory into an ASCII database format and save them to the hard disk.
- to convert a file saved in ASCII format into a PADS-2000 database or part of a database, and load it into the system memory.
- to input a file that is in a FutureNet™ netlist format into a PADS-2000 database.
- to convert a file saved in Racal Redac MAXI/Redboard™ ASCII format into a PADS-2000 database, and load it into the system memory.
- to input a netlist file that has been created by Schema™ schematic capture software.
- to create a new layout without starting from a netlist (so-called "Design-on-the-Fly").
- to erase a design from memory and the CRT without leaving the design program.

3.1 Binary Job Vs. ASCII File

The I/O menu handles two types of file formats for PADS design files. These are Binary Job files and ASCII format data files.

The binary file is the standard PADS database format. Normally, you will save and load your databases in binary files. These files are called "JOBs". A "Job" applies to PADS-2000 Printed Circuit Board databases. The binary format is unreadable by the designer and is not editable except graphically in PADS-2000. Job files are identified by their file name and the extension .JOB, e.g. MYJOB.JOB.

ASCII files are human-readable text files containing information on the design. The ASCII format can be brought up in the PEDIT text editor and viewed or changed. ASCII files are identified by the file name followed by the extension .ASC. The ASCII database format has been provided to enable users of PADS-2000 to extract information from a design database for use in other manufacturing or engineering processes without having to understand the PADS binary file. Data can also be added to a design file in ASCII format as well. The entire database is defined in the ASCII file.

The format of a PADS-2000 ASCII file is provided in Chapter 13 of this manual. Users of PADS-2000 have the advantage of "open architectural software" in that they can interface systems both "to" and "from" the PADS-2000 database.

A JOB file cannot be added to existing Binary design data currently in memory. ASCII job data can be added to existing Binary data currently in memory.

The data contained in a JOB file will not be checked for accuracy while being recalled to memory. ASCII data IS checked as it is being read in.

All files, both .JOB and ASCII are stored in a subdirectory of PADS called FILES.

3.2 IN/OUT Menu

To enter the IN/OUT menu, access the Major Command Menu. The hierarchy of menus must be displayed as shown in Figure 2.1. Note that the IN/OUT menu is the first major menu. It is selected by: (1) selecting the F1 function button or (2) positioning the highlight bar over 1. IN/OUT and pressing the mouse #1 button. When selected, the major menus are replaced with the In/Out submenus shown in the following Figure 3.1:

1 Job In 2 Job Out 3 Ascii In 4 Ascii Out 5 FutNet In
6 Conv In 7 Schema In 8 On-The-Fly 9 Erase Job 10 Exit

Figure 3.1 - THE IN/OUT Menu

To exit the IN/OUT menus and return back to the Major Commands menu, use the F10, EXIT, function button.

3.3 Loading a Design into PADS-2000

The JOB IN (F1) command loads into memory and displays it on the CRT, a design previously stored on the hard disk by the JOB OUT command. The communications window located at the bottom of the CRT asks the question:

JOB INPUT FILENAME (CR=PCB.JOB):

Respond by typing in the name of the JOB file that is to be recalled, followed by <CR>. (To recall the Backup or Security file, simply strike the <CR> key.)

You may also select a list of potential files by typing:

*** <CR>**

A pop-up window containing an alphabetical listing of files in the \PADS\FILES directory appears. Use the mouse to position the cursor over the name of the job file to be input and press mouse button #1. The named file will be inputted.

NOTE 1: A file from a directory other than \PADS\FILES may be selected by specifying the entire search path,. You may also read files from floppy disk by specifying drive A:

NOTE 2: If more files are in the selected directory than will fit in the pop-up window, you may cycle thru the pop-up window by selecting MORE at the bottom pop-up window with the mouse cursor.

The message "WORKING..." is displayed in the Information Status Window and the design is moved into memory and displayed at the CRT after some time.

If the job file requested is not on the disk, the error message:

JOB FILE NOT FOUND

will be displayed in the communications window.

If a design is currently in memory and displayed at the CRT, it will be overwritten by the new JOB file recalled from the hard disk.

3.4 Storing a Design to Disk

The JOB OUT command is used to store the current active design data onto the hard disk. After a design is stored on the hard disk, it also remains in memory, and continues to be displayed.

The JOB OUT command is activated by selecting Function Key F2 in the In/Out menu. When selected, the message:

JOB OUTPUT FILE NAME (CR=PCB.JOB)

appears in the prompt line. Type in the desired name of the output file followed by <CR>. The file will be written to the hard disk. The CPU message window displays "WORKING..." for a few seconds, indicating that the job is being stored to the disk. Make a practice of watching for this message immediately after striking <CR>. This is your assurance that your design is being stored.

The name of the design file can be any valid DOS file name that is 1 to 8 characters in length. Use all Alpha and Numeric characters, as well as the following special characters: !, @, #, $, &, _, {,%, }, ~, ` . We recommend, however, that special characters not be used unless absolutely necessary.

NOTE: A space or a tab is NOT accepted in the Job Name.

To store the design as a "Backup" file, simply use the Carriage Return key. The prompt:

OVERWRITE FILE Y/N?

will appear in the communications window. If you wish to overwrite the existing stored Backup job file, respond with "Y"; the stored Backup design will be destroyed by being overwritten with the current design.

If you do not want to overwrite the existing job, respond with "N" and provide a different name to store the job.

NOTE 1:. A file created with the JOB OUT command will have the extension .JOB automatically assigned to the file name.

NOTE 2:. You must be aware of the difference of storing/outputting versus loading/inputting. The danger is losing the contents of a job in which you have invested many hours of your labor. REMEMBER THE FOLLOWING: JOB IN means to Bring IN stored information FROM the hard disk to the CRT and memory. JOB OUT means to move data from the memory and CRT, OUT to the hard disk. The classical error is for a designer to work on a board for several hours and at the end of the shift, store the job (he thinks). In the process, instead of selecting the JOB OUT command, the designer inadvertently selects JOB IN. The unfortunate results are recognized on the next shift when the response:

JOB NOT FOUND

is given when trying to Recall the job that was thought to be stored.

NOTE 3: A Suggestion: When intending to store a job, think "I want to move and store the file from memory and CRT OUT to the disk," therefore, use JOB OUT. Secondly, when wanting to recall a design, think "I want to recall from disk and load IN memory," therefore, use JOB IN. When making a JOB OUT, closely observe that the CPU message window shows "WORKING..." for a few seconds. If you want to be VERY sure that you have stored your design, before you power down the CPU, simply recall the design that you have just stored using the JOB IN command. If it "Recalls," then it has been stored properly.

NOTE 4: You will be relieved to know that if you have inadvertently made an error in storing jobs, the PADS-2000 Program has a unique data security feature called "Backup" that allows you to take corrective action if you discover your error in a timely fashion. A file backup of the current design is made regularly, and this may be recalled by selecting JOB IN and typing <CR>. This Backup feature is discussed in Section 4.4.10.

3.5 Reading an ASCII File

The ASCII IN command is used to read an ASCII file from the disk, convert the ASCII file into a PADS binary format, and to load the data into the current active design in the memory. An ASCII file can consist of all or part of a PCB design, including the board outline, part information, netlist data, and standard system defaults. Any information which is definable in PADS-2000 and can be stored with a JOB file can be defined with the PADS-2000 ASCII database.

ASCII Input files can originate from three sources: (1)You create it yourself from a manually drawn schematic; (2) From PADS-Logic, (3) From other schematic package that has a PADS netlist format. A complete discussion of ASCII files, their content and format is contained in Chapter 13, ASCII Files Format.

The ASCII IN command is activated by the selection of the F3 Key. Respond to the message:

ASCII INPUT FILE NAME (CR=BACKUP.ASC):

with the name of the ASCII file, followed by <CR>, or with

*** <CR>**

to see a pop-up window displaying all of the ASCII files in the directory. If the latter, you will then select the desired file from the window with the cursor.

After selecting the input file, you will be requested with the name of a file in which to record errors. To the prompt:

ERRORS OUTPUT FILE NAME:

provide a name for a file which will contain any errors followed by <CR>, or type

<CR>

to get a hardcopy printout of errors.

When using this command, be aware of the following:

• The ASCII file is automatically converted to a Binary format when recalled into memory.
• The ASCII file is added to an existing JOB file already in memory and displayed at the CRT.

In the process of reading in an ASCII file, extensive data checks are made on the data in the file to eliminate errors before the conversion to a JOB file. In most instances, an input data line with an error will not be converted until the offending error is corrected. The error checking consists of "Logical" errors such as: pad numbers in the connection list that do not exist on the referenced part, reference designations in the connection list that do not exist in the parts list, and similar "logical" errors.

Any input file errors can be sent to a file for later analysis by responding to the request for the error file name message with the file name. Alternately, any errors can be sent directly to the printer by responding to the Error Message requesting a file name, by striking the Enter <CR> Key. We suggest the preferred procedure is to send the error file directly to the printer.

A large ASCII file will take several minutes to be read in. During read in,

"WORKING..."

is displayed in the CRT Window.

3.6 Creating a PADS ASCII File

This command is used to convert an existing design in memory into an ASCII file and to store this file on hard disk or print it. To implement the ASCII OUT command, select the F4 Function Key. A pop-up window as shown in Figure 3.2, appears in the viewing area.

By responding YES, data is output in the current unit of measure, as the User had previously defined in the SETUP, UNITS menu, i.e., Metric, Mils or Inches.

By responding NO, data is output in System Database units, the most precise unit of measure. Be aware that outputting in a grid other than the basic system grid will produce round off errors if you intend to convert from inch to metric or metric to inch. This is because some degree of database precision is lost when you do not output the database in the basic system grid. Therefore, the user is advised when precise accuracy is required, output in Basic units.

The generation of ASCII OUT for a large circuit board will take a few minutes to complete. The word "WORKING..." will appear in the CRT Window. The same rules for naming a JOB OUT file apply to naming of the ASCII OUT file. The extension .ASC is automatically assigned to an ASCII stored file.

ASCII OUT files can be used in the following manner:

(1) You can obtain a printout of the contents of the ASCII file and examine it in greater detail than when this file is graphically displayed at the CRT.

(2) Some changes can be better implemented using the PEDIT text editor, to edit the ASCII file then in PADS-2000.

(3) The ASCII file can be used by your programmers to convert to a file created in PADS-2000 with other computers, programs or activities in your facility.

(4) The ASCII Out/In feature is an excellent database diagnostic tool. Should database items ever become contaminated as a result of hardware or software bugs, make an ASCII Out file of the suspected file, erase the contaminated file from memory, and read the ASCII file into memory using the ASCII IN feature. The ASCII In data checking functions will usually identify the contamination, allowing you to delete it and replace with good data.

3.7 Loading a FutureNet Net List

It is possible to read a netlist file created in FutureNet format directly into PADS-2000. This allows users of FutureNet and other schematic programs that generate FutureNet net lists to design their circuit boards with PADS-2000.

To use this, first place the FutureNet file in the PADS subdirectory called FILES, so that it can be accessed by the In/Out menu.

Next, create a printout of the file and study it for areas that may require changes. For example, the Part field (Attribute 3) MUST correspond with the Part Type name (not Part Decal name), in the PADS Library. If there are discrepancies, either change the schematic or change the PADS Part Type name file.

Signal Net names in the FutureNet file can be up to 12 characters in length, while Part Type names may be up to 16 characters in length. Alpha and numeric characters are legal. The following are NOT legal and are not accepted: a space, { } < > ( ) [ ] ! ~ @ % ^ & . - : ?

In the FutureNet file, insure that all I/O connectors have Reference Designations and Part Types. In fact, ALL parts must have Reference Designations.

Reference Designations may have up to six (6) alphanumeric characters and may not end in an alpha character. Alpha characters at end of Ref Des will be deleted. Part Types may have up to 16 Alphanumeric characters.

Using the Editor called PEDIT, make these changes to the FutureNet file and place the edited file back in FILES. Use a file name with the extension .Net, e.g. SAMPLE.NET.

Before reading the FutureNet file into PADS, it is suggested that you have the Board Outline in memory and displayed, and have the System Origin set to the Board Outline origin. When the FutureNet file is read into memory and the Part Decals are brought up and displayed, they will appear at the 0,0 location (lower left corner) of the PCB. If you do not have the Board Outline in memory, at least move the system origin up away from the lower left corner of the CRT, to some location in the

working area where the parts can be viewed, after being read in and prior to placement.

Select the menu In/Out. Select F5, FUTNET IN. In response to the message:

FUTNET INPUT FILE NAME (CR=FUTURE.NET):

enter the FutureNet file name. In response to the message:

ERRORS OUTPUT FILE NAME:

provide a name for a file which will contain any errors found when reading in the FutureNet file. Alternatively, to get a hardcopy printout only of errors, use the Enter <CR> key, rather than a file name.

FUTURENET IN (F5)
FutureNet Input File Name (CR=FUTURE.NET)
Errors Output File Name (CR=PRINTER)
Strip Out Gate Info and Duplicate Parts From Input File Y/N?

If you have specified UIA, UIB, etc., or common pins to packages (Sips, enables on IC's, etc) in your FutureNet Schematic you would answer "Y" to strip out the A,B,C, etc., since parts UIA, UIB, etc. are seen as duplicate parts by PADS.

Part Type Attribute [3]>
Part Value Attribute [4]>
Part Tolerance Attribute [6]>

If the user only requires one attribute and not the others, they must specify 0 for those not required. If the user wants to specify a FutureNet "user definable attribute" they may; however, the user defined attribute must appear in the file in the same location as either attribute 3, 4 or 6.

Not Connected Pins Net Name>

If a [CR] is entered it is assumed that SIG,,,,, entries in the FutureNet file are the no-connects, otherwise the user must specify the no-connect sig name used.

If the user specifies a no-connect Signal Name, AND there is a "SIG,,,,," in the file, "PADS" interprets this as bad line data.

It is not a problem because the pins listed under SIG,,,,, ARE no-connects. It is recommended the user check this section of the FutureNet file to the Schematic and insure they are truly no-connects.

Once the FutureNet is read in and complete, one of two messages will appear:

INPUT ERRORS FOUND

indicates there is a problem with either the FutureNet file, or there is data missing from the PADS Library. Check the Error Output File for a clue.

NO ERRORS FOUND

indicates there are no errors, however, there COULD BE Warnings! Check the Error Output File.

In either case -- check the ERROR OUTPUT FILE!!

A sample FutureNet NetList, together with comments is shown in the following:

SAMPLE FUTURE NET FILE	REMARKS	(below list)
NETLIST,2	See R1	
(DRAWING, FUTURE1.DWG, 1-1 DATA, 54, September 21, 1987)	See R2	
(DRAWING, FUTURE1.DWG, 1-2 DATA, 54, September 21, 1987)	See R2	
(SYM,1-1,2	See R3	
DATA,2,U1	See R3.a	
DATA,3,7400	See R3.b	
DATA,25,1		
DATA,25,2		
DATA,25,3		
etc		
-		
etc		
-		
etc		
)		

For Discrete Parts such as Resistors, Caps, etc., where the part name/description may include characteristics of the part such as resistance value, tolerance, etc., and where the inclusion of such data in the PADS Part Type name would result in many thousands of parts in the PADS Library, a feature called Part Family and Tags is available in PADS which in conjunction with the FutureNet Attribute concept, will greatly simplify the Part Type identification. This feature and its use in FutureNet is described below:

A Discrete Part Family as identified by FutureNet Attribute 3, as a family or category of parts. Carbon resistors for example, are identified by family according to wattage, i.e., R1/4W is a quarter watt resistor, R1W is a one watt resistor; all quarter watt resistors use the same Part Decal - one watt resistors use the same Part Decal, different of course from one quarter watt resistors. The FutureNet Attribute 3 would thus record:

DATA,3,R1/4W

The PADS Part Type name would be R1/4W. In addition, the PADS Part Decal name would also be R1/4W. The first line of the part type data in the PADS Library would therefore read:

```
R1/4W  R1/4W RES 000000
```

(See Volume II, Chapter 12, System Parts Library for complete details of the Part Type data).

With the PADS-2000 Library system, additional data such as VALUE and TOLERANCE can also be handled. The FutureNet Attributes 4 and 6 are used to add VALUE and TOLERANCE Tags to the Part Type family name.

Examples are as follows:

```
DATA, 4, 1K          1K and 5% are Tags
DATA,6, 5%
```

In PADS-2000 the Part Family Name (R1/4W) is used to call up the Part Type and its associated Part Decal (a quarter watt resistor) from the library. AFTER extraction of the Part and Part Decal from the PADS Library and upon entry of the part into the board database, the VALUE and TOLERANCE "tags" are added to the Part Family so to uniquely describe the part. Once described in the database, all usages of the part type such as ASCII OUT, Parts Lists, etc., will identify the part described as R1/4W, 1K, 5%

CAUTION: The sum total of all characters from the Part Family and VALUE and TOLERANCE Tag attributes CANNOT exceed 16 characters, including Alphanumeric and special characters, such as commas and spaces. (PADS inserts commas to distinguish between Part, Value, Tolerance).

The following is a partial FutureNet listing of three different parts that illustrate both Digital and Discrete parts, including part family and tags.

```
DATA,2,C7
DATA,2,C7        <------ ATTRIBUTE 2  =  REF DES
DATA,3,CK05      <------ ATTRIBUTE 3 = PART FAMILY
DATA,4,0.1UF     <------ ATTRIBUTE 4  =  VALUE
DATA,118,L1324A00,<--USER DEFINABLE ATTRIBUTE, IN
                 THIS CASE, A COMPANY PART NUMBER.
DATA,1,1         <------ ATTRIBUTE 1=  PIN NUMBER, PIN 1
DATA,1,2         <------ ATTRIBUTE 1 = PIN NUMBER, PIN 2
)
(SYM,1-2,192     <------ SYMBOL,SET 1-SHEET 2, SYMBOL #192
DATA,2,U19       <------ ATTRIBUTE 2 = REF DES
DATA,3,7400      <------ ATTRIBUTE 3 = PART
DATA,118,L1854A00<---USER DEFINABLE...
DATA,23,1        <------ ATTRIBUTE 23 = INPUT PIN, PIN 1
DATA,23,2        <------ ATTRIBUTE 23 = INPUT PIN, PIN 2
DATA,21,3        <------ ATTRIBUTE 21 = OUTPUT PIN, PIN 3
)
(SYM,1-1,196
DATA,2,R1        <------ REF DES
DATA,3,RES400 <------ ATTRIBUTE 3 = PART
DATA,4,1K        <------ ATTRIBUTE 4 = VALUE
DATA,6,5%        <------ ATTRIBUTE 6 = TOLERANCE
DATA,118,R22654R00<--USER DEFINABLE...
DATA,1,1
DATA,1,2
)
```

At the end of the SYM portion of the FutureNet File, PADS has received all the Parts and their Reference Designations, accessed the PADS Library and created a board database of parts including Decals. Following this, the connection part of the FutureNet Netlist is entered as per the connections or NetList portion. A sample FutureNet Netlist file of connections is as follows:

```
(SIG,,***132114,1-1,,   See R4, R4.a,        R4.c
PIN,1-1,2,U1,25,1       See R4.b
PIN,1-1,1,U6,25,1       See R4.b
)
(SIG,,***132118,1-1,,
PIN,1-1,2,J1,25,2
PIN,1-1,1,J3,25,2
)
```

```
(SIG,,***123122,1-1,,
PIN,1-1,2,J1,25,3
PIN,1-1,1,J3,25,3
)
(SIG,,***123126,1-1,,
PIN,1-1,2,J1,25,4
PIN,1-1,1,J3,25,4
)
etc
```

REMARKS Pertaining to sample Future Net File (above)

1. Must have NETLIST, 2 header

2. Information from Title Block, Sheet 1 Data, 54, September 21, 1987.

Information from Title Block, Sheet 2 Data, 54, September 21, 1987.

All Drawing items are ignored by PADS-2000.

3. SYM is the key word for Part Type data.
- Last field of Attribute 2 [(SYM] is Reference Designation: U1
- Last field of Attribute 3 [(SYM] is Part Type Name as contained in the PADS Library: i.e., 7400. These Part Type names must match the ones in the PADS-2000 Library. If not, editing is required. The remaining data associated with (SYM is ignored.

4. SIG is the key word for connections data.
- Second field of [(SIG is the Signal Name. PADS-2000 requires a Signal Name.
- Fourth and Sixth fields of attribute 25 lines [(SIG] is the Net Node. For example Signal Name 132114 contains the net U1.1, U6.1..
- Second field (which would be between ,) of Line 1 [(SIG] is the Bus Signal Name. Bus Signal Names are handled, for example, as follows:

(SIG,DATAIN, 00.....	translates into	DATAIN00
(SIG,DATAIN, 01.....	translates into	DATAIN01
(SIG,DATAIN, 02.....	translates into	DATAIN02

If you insure that the above criteria are met in your FutureNet Out file, then conversion to PADS-2000 via the FutureNet In menu should be processed without difficulty.

A listing of possible errors messages, and their cause is provided as follows:

101 CAN'T FIND PART DECAL ITEM <D1P14)
*Called by part type <7410>
Part package not found -2
(SYM, 1-1, 3
DATA, 2, U1
DATA, 3, 7410
***** CAN'T FIND THE PHYSICAL COMPONENT IN THE LIBRARY (DECAL) TO BE PLACED ON THE PC BOARD. (THE DECAL IS REFERENCED IN THE PADS PART TYPE FILE AS DIP14).

101 CAN'T FIND PART TYPE ITEM <7410>
(SYM, 1-1, 3
DATA, 2, U1
DATA, 3, 7410
***** CAN'T FIND THE PART TYPE FILE FOR A 7410 IN THE PADS PART TYPE LIBRARY NOTE: IN THE PROCEEDING ERROR MESSAGE, THE PART DECAL COULD NOT BE FOUND IN THE PADS LIBRARY. IN THE ABOVE EXAMPLE, THE PART TYPE COULD NOT BE FOUND. A SUBTLE BUT IMPORTANT DIFFERENCE, INDEED!

PART NAME ALREADY USED U1
(SYM, 1-1, 3
DATA, 2, U1
DATA, 3, 7410
***** REFERENCE DESIGNATOR U1 IS CALLED FOR IN THE SCHEMATIC MORE THAN ONCE. (AS A GATE) ANSWER "Y" TO STRIP OUT GATING INFORMATION AND DUPLICATE PINS FROM INPUT FILE.

BAD PIN NUMBER IN CONNECTION R1 3
***** COMPONENT R1 HAS MORE PINS THAN THE PHYSICAL DECAL HAS DEFINED. R1 IS A 2 PIN RESISTOR, PIN 3 IS BEING DEFINED IN THE ORIGINAL INPUT SCHEMATIC.

ADDING DUPLICATE CONNECTION U1 14 R1 1
PIN, 1-2, 10, R1, 1,1
***** A COMPONENT PIN NUMBER IS DEFINED MORE THAN ONCE IN THE SCHEMATIC. IN THIS CASE, R1 IS A SIP PKG, AND PIN 1 IS SHOWN TWICE IN 2 SYMBOLS (R1 PINS 1 & 2, R1 PINS 1 & 3), BOTH OF WHICH ARE TIED TO +5V. THIS IS A WARNING, AND THE USER MUST VERIFY IT IS THE SITUATION DESIRED. IF THEY ARE DIFFERENT SIG NAMES, THE LATTER OF THE DUPLICATE SIGNAL NAMES WILL BE DROPPED.

* BAD FUTURENET SIG DATA LINE FORMAT
(SIG,,,,,
***** THIS IS THE START OF THE FUTURENET UNCONNECTED PINS SECTION. THIS CAN BE TAKEN AS A WARNING MESSAGE HOWEVER; IF THE USER HAS DASH4 REV 2.45A OR EARLIER, ACCEPT THE WARNING MESSAGE AND MAKE SURE TO RUN A "NETLIST" REPORT ON PADS TO VERIFY THE CONTINUITY BETWEEN THE FUTURENET NETLIST AND THE PADS DATABASE; THE PADS "UNUSED" REPORT SHOULD REPRESENT THE SAME DATA AS THE SCHEMATIC.

IF THE USER HAS DASH4 REV 4, THERE IS A NETLIST PROCESSOR THAT ENABLES THE USER TO HAVE "DASH" DROP THE UN-CONNECTS, THUS THE "SIG,,,,," STATEMENT SHOULD NEVER APPEAR IN THE FILE.

3.8 Loading an Orcad Netlist

Users of the Orcad SDT can use the FutureNet output from Orcad if the following utility is used in Orcad:

NetList Input.sch Output.net FutureNet /S/O/N/P

where
NetList is the name of the utility
Input.sch is the name of the input schematic
Output.Net is the name of the netlist
S = Future Net

O = (letter O, not zero) is for a one page schematic, higher numbers for multi-sheet schematics.
N = NetList
P = use pin numbers, not Alpha characters

3.9 Converting a Redac Design to PADS-2000 Format

The CONV IN command allows the acceptance of a REDAC ASCII OUT file from the Redac Maxi™ and Redboard™ series. Off grid data is accepted and converted to the PADS-2000 0.00001" data base. In the process of making the conversion you should be aware of the following:

1. Board outline and component outline are drawn with 10 mil lines when entered into Pads. They may be changed if desired.

2. REDAC does not have part names, instead library items are used. (e.g. L25)

3. Trailing blanks will be deleted from TEX strings. (Some REDAC users use trailing blanks to make box larger. Since PADS doesn't display blanks, they are useless). Entirely blank text strings are deleted.

4. Unrouted connections use largest of widths for selected COD. (REDAC supports separate widths for each layer. PADS does not). For example, if COD 2 widths are 10 for top, 12 for bottom, 0 for inner, then PADS uses 12.

5. Plated-through-holes indicator in REDAC Pad codes is ignored.

6. NeckDown is ignored.

7. Bright-up setting is ignored.

8. PADS does not support more than one set of spacing rules. Largest space rule for any layer in the Redac database is used. For example, if Track-to-Track spacing is set to .010" on inner layers and .012" on outer layers, then .012" will be used.

9. Off-grid components are handled by being attached to the true (off-grid) pads. This may cause some spacing violations but they will be easy for the user to repair.

10. PADS-2000 selects the REDAC board outline with the most corners as the true outline. REDAC supports multiple board outlines. PADS-2000 selects only one: The one with the most corners.

11. The .ASS keyword CMD is ignored.

12. PADS-2000 does not support fractional grid sizes. Therefore, REDAC DSU values which specify fractional grids (e.g. 60 or 80 DSU) are not accepted.

Place the Redac file to be converted in the subdirectory called FILES. Next, make a print out and study it for any changes that will be necessary. For example Redac Library items use the format LXXX. These will need to be changed to the PADS-2000 Library Names using the PEDIT editor.

Use a file name with the extension .PCI. Select In/Out Menu. Select F6, CONV IN. In response to the message:

CONV INPUT FILE NAME (CR = REDAC IN)

enter the Redac file name. In response to the message:

ERRORS OUTPUT FILE NAME

provide a name for a file which will contain any errors found when reading in the Redac In file. Alternately, to get a hardcopy print out of errors only, use the Enter <CR> key rather than a file name.

3.9.1 Meaning of Error Codes in Redac Conversion

Error Codes produced by REDAC Interface

1xxx	Any Section
1001	Line too long or illegal character
1002	Unrecognized "." keyword
1004	Expecting "." keyword

8xxx		COP Section
	8002	Missing coordinate
	8030	Last copper shade not terminated
	8100	Error adding cooper to database
9xxx		TEX Section
	9020	Code width missing
	9100	Missing field
	9101	Error adding text to database
	9320	Bad layer number
10xxx		OFF Section
	10010	More pins than expected
	10012	Too many pins
	10015	Fewer pins than expected
	10019	Unexpected line
	10100	Error reading pad code assignments
	10301	Error modifying symbol
	10401	Error reading package from database
	10435	Error reading symbol from database
	10712	Missing pin number
	10714	Missing Field
11xxx		IDX Section
	11010	Unrecognized line
	11015	L number out-of-range
	11020	Unrecognized flag

3.10 Schema In

The Schema In command (F7) permits users to input netlists created by the Schema Logic Capture System, containing parts and connections. The Netlist format used by Schema In is in FutureNet format.

Select In/Out Menu. Select SCHEMA IN (F7). Respond to the message:

SCHEMA INPUT FILE NAME (i.e., SCHEMA.FNN)

enter the Schema file name. In response to the message:

ERRORS OUTPUT FILE NAME:

provide a name for a file which will contain any errors found when reading in the Schema In file. Alternately, to get a hardcopy printout of errors only, use the Enter <CR> key rather than a file name.

Follow the same interactive question and answer sequence as for FutureNet - Section 3.7.

When utilizing pin names in from SCHEMA, (E=emitter, B=base) PADS-2000 runs a conversion program. The conversion is as follows:

C G L T C = pin 1
B D R B A = pin 2
E S = pin 3

The abbreviations are as follows: Collector, Base, Emitter, Gate, Drain, Source, Left, Right, Top, Bottom Cathode, Anode.

3.11 On-The-Fly

The command of On-The-Fly permits the creation of a design without having to start from a netlist input. It is generally used for the design of smaller boards where the schematic either does not exist or is in the process of being generated and the engineer desires to proceed with the design of the board without waiting for the part and net list to be generated.

When (F8) On-The-Fly is selected the menu changes as shown in Figure 3.3 below.

1 Add Conn 2 Del Conn 3 Ren Net 4 Disconn 5 Add Part
6 Del Part 7 Ren Part 8 Mod Rut 9 Add Rut 10 Exit

Figure 3.3 On-The-Fly

The command options of On-the Fly are as follows:

3.11.1 Adding a Connection with On-the-Fly

ADD CONN permits the adding of a Connection to the existing board database in memory.

To add a Connection to the board:

1. Set WIDTH to the desired new Track Width with the W modeless command.

2. Select ADD CONN. If the mouse is to be used to enter the connection, place the cursor over the correct part pin and select 1, SELECT.

3. The net starts at the pin over which the cursor is located and the menu changes to that as shown in Figure 3.4.

1 Select	**2 Keyboard IO**	**3 Undo**	**4 Ren Net**	**5 Window**
6 Expand	**7 Contract**	**8**	**9**	**10 Exit**

Figure 3.4 - ADD CONNECTION MENU

The rubberband connection follows the movement of the cursor. Move the cursor to the next component pin. Connect the connection to the pin by choosing SELECT (F1). If the wrong pin was selected, use F3, UNDO, to disconnect the last pin. Continue to move the cursor to new pins until the entire net is added, then complete the net by using F10, EXIT.

When the connection was started the system gave a signal name to the connection as displayed in the Information pop-up window as $$$nnn where $$$ indicates the net name was put in by the system, and nnn is an arbitrary netname. If you wish to change the net name, select (F4) REN NET and respond accordingly.

If the keyboard is to be used to type the connected pins, select KEYBOARD I/O (F2). The message center provides the message:

STARTING CONNECTION PIN -
REFERENCE DESIGNATOR.PIN>

Enter the Reference Designator of the part and the pin number or name of the start point of the connection, separated by a period (.).

The name of the net is given in the pop-up Information window, the cursor moves to the pin and Command Window changes to Figure 3.5, ready for the next pin to be entered. Keyboard I/O F2 will enter the connection at the pin and the following message is given.

NEXT CONNECTION PIN - REFDES.PIN >

Type the next part and pin for the connection .

In either method of entry, mouse or keyboard, the end of the net is made by F10, EXIT.

On-Line Error Messages are generated in the command to eliminate errors while adding connections. Some are as follows:

- SIGNAL NAME ALREADY USED
- ADDING DUPLICATE CONNECTION NODE 1 NODE 2
- CONNECTION TIES NET X INTO NET Y
  followed by message:

NAME FOR COMBINED NET Z

From time-to-time, use the Length Minimization command of PLACE to minimize the length of a connection net. After using this command, create a Net List and Part List and review for accuracy. If you find you need to delete a connection, use DEL CONN.

3.11.2 Deleting a Connection with On-the-Fly

DEL CONN is used to remove a connection or track from the board displayed at the CRT. To delete a connection:

1. Select DEL CONN. In response to the prompt, place the cursor over the track or connection and SELECT. The connection will disappear from the CRT (but not from memory). The connection identification will appear in the Information Window and the message:

CONFIRM DELETE Y/(N)?

4.1.5 Displaying Parts

Parts that are stored on the Top layer of the circuit are distinguished as a category from those on the bottom. The display control for the decal outline of Parts on the top is done with the Top option, while those on the bottom are controlled with Bottom. The first entry in both Top and Bottom is E, for Electrical. This determines the color to be used for displaying the Part outline for parts on the top (where E is always level 1) and for part outlines for Parts on the bottom (where E is equal to the Max Layer selection, which is the Bottom layer of the board). Note that the bottom layer is set invisible in both entries.

PADS-2000 distinguishes between the representation of Top level Parts and Bottom level Parts to enable users to display decal outlines created on other layers, for example on the silk-screen layer, as different colors.

Part Name is the Reference Designation (REF DES) of the part as used in the schematic and parts list. Part Type is the Part Type name, i.e., 7400, 7404, etc. Part names and Part Types are selectable only on layer 1 (the component layer) and the layer you have selected as the Max Layer for routing, which is considered the bottom or solder layer. This is because parts are allowed on only the component and solder layers. (see section on Max Layer).

The Silkscreen component outline can also be displayed, if the Silkscreen component outline was created on the Silkscreen Layer (LY 26), when the component Decal was originally made. To display the Silkscreen component outline, in addition to the Assembly Drawing Component outline, use DISPLAY and set 2-D Line visible on all layers of artwork, as well as the Silkscreen layer (Layer 26). The Silkscreen component outline will be displayed in the same color used for PART SHAPE.

4.1.6 Connections

Connections are the rubber band "to-from" connections between part pins before they are converted into Tracks using the Routing menu. For connections to be displayed, even if set Visible, they MUST be turned on using the NET ATTR DISPLAY ON discussed in Section 4.5.

The 0 level indicates that connections will be shown on all levels when made "visible." Connections are not associated with a specific layer in the database.

4.2 Defining Pad Stack Sizes

The PADS command is used to change the size/shape/drill hole size of selected pads of Parts and Vias in the current active board board design.

Each pad, pin or finger of a Part Decal has a Pad Stack associated with it. Each Pad Stack has a maximum of 30 levels which corresponds to the 30 levels within the database. Each level of each Pad Stack can be set to a different pad size and shape, if required.

When a part decal is accessed from the library, it includes the Pad/Pin Numbers, Pad Sizes and Shapes established during the creation of the Part Decal and stored with the part. You may wish to change the pad information for a part in the board under design. The Pads command of the Setup menu is used to change the original Pad Stack data of a Part Decal in the current design only. When a Part Decal pad stack is changed in PADS-2000 using the Pads command, it is changed for that particular board only; the Part Decal in the System Library is not changed.

NOTE: If you wish to change the Pad Stack data for a Part Decal that is in the Part Decal Library as well as the board database, you must use the Create Part Decal command described in Chapter 5, followed by changing the part in the board database as described in Section 9.1.1.

To change a Pad Stack of a Part on a board, you select Pads, F2. The message in the prompt line is:

NAME OF PART DECAL >

Type the name of the part decal in the design whose pad stack you want to modify. If the Part Decal is in the design, a pop-up window appears, listing the pad stack information for all pads in the decal. An example appears in Figure 4.3.

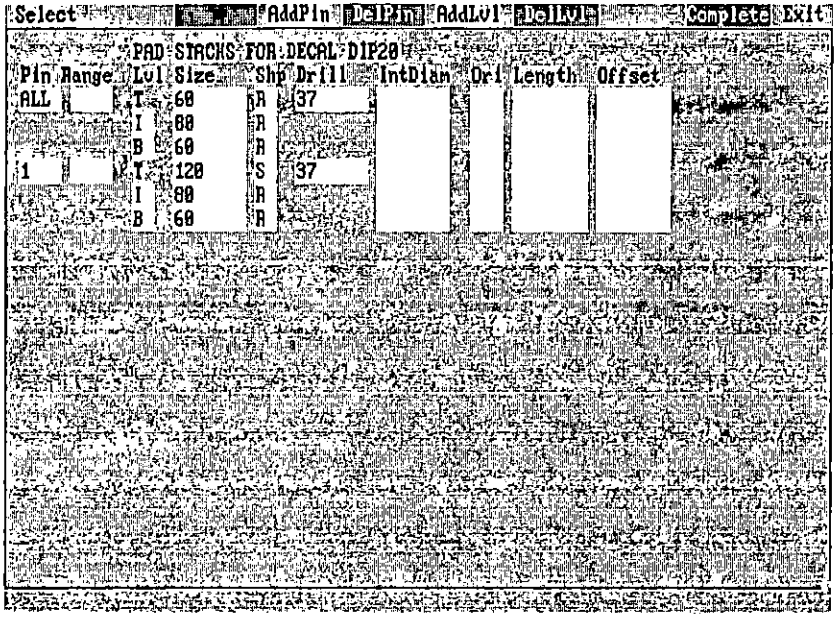


Figure 4.3 - The Pad Stack Form for a DIP20

This form gives the current padstack definition for the Part Decal. There is an entry for each unique padstack in the part. A pad stack may be shared by one or more pins, so there may well be fewer padstacks than pins in the Part Decal

The first two columns list the pin number(s) being defined. This may be a unique pin number, a list of pin numbers, a range of pin numbers, Even for all Even number pins, Odd for all Odd pin numbers, or All, which defines the padstack to be used for for all pins not explicitly specified. The first line entry is for All pads, i.e. for all pads not otherwise described. Then, for each unique pad stack configuration that is used for one or more pins, there will be an additional entry.

The next column entry defines the levels being described. Each pad entry has a unique description for at least two levels, Top and Bottom. Multilayer designs have at least one additional level description for Inner layers. Up to 30 unique levels can be described for the pad stack.

For each pad entry, there is a column giving the definition of the pad size, pad shape shape, and drill hole. Additional information may also be specified, depending on the pad shape. The information described is as follows:

Size - enter a value in the current system units, depending on the pad shape: round or annular pad, the diameter; square pad, the length of a side; oval or rectangular finger pad, the width.

Shape - enter shape using the following codes:

R = Round pad
S = Square pad
A = Annular pad
OF = Oval finger
RF = Rectangular finger
O = Odd-shaped pad. Used for such items as Moire (registration) pads, where the internal contents of this pad are not significant. Instead the shape and overall size are needed for DRC checking. Odd shaped pads are mapped to a special aperture in the photoplotter aperture wheel, as are Thermal Relief pads.

Drill - enter Drill Hole size in the current dimension unit. An entry of 0 will result in no drill hole.

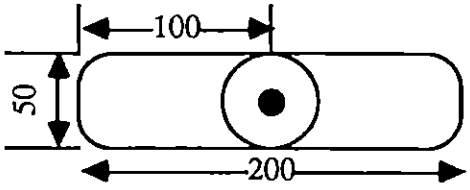
If the shape value was OF or RF, you must add additional information for Orientation, Length, and Offset.

Orientation - enter Orientation of Finger with respect to Part Decal orientation during construction, in degrees in the range 0 to 179.

Length - End-to-End Length of Finger in the current system unit.

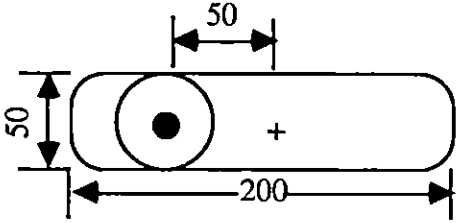
Offset is used to SHIFT the finger relative to the Electrical Center of the Pad Location on the Part Decal. It does not change the shape or extend the length of the pad. A positive offset shifts the finger up or to the right the defined distance. A negative offset value shifts the finger down or to the left the defined value. The maximum offset is ± 1.000", and is not to exceed one-half the length of the finger.

E.G., a .2" x .050" finger thus:

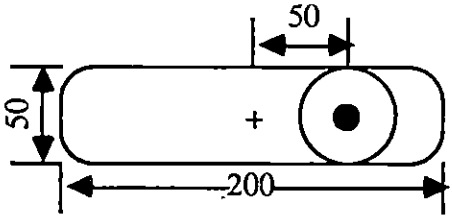


● = Electrical Center

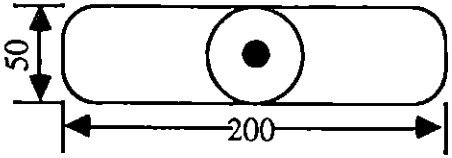
If we give a .050" offset, we offset the finger relative to the electrical center by .050" thus:



If we give a -.050" offset, we offset the finger relative to the electrical center by -.050" thus:



In order to maintain electrical continuity with the drill hole after the finger is shifted, Design Rule Checking would preclude offsetting the pad to a degree that the original drill hole and the shifted finger were disconnected. Such a case is shown in the example below.



A .100" shift would result in the following:

This would result in the Error Message:

FINGER OFFSET IS OUTSIDE THE PAD

Annular pads are pads that have a clearance contained within the pad. The entry of A, to the SHAPE defines the pad stack to be Annular.

When Annular pad shapes are selected, you must define the diameter of the inner circle in the column IntDiam., in current system units. The result will be a pad whose outer diameter will be established by the SIZE entry and whose inner diameter is established by the IntDiam entry

4.2.1 Adding Padstack Definitions

Additional Padstacks can be defined by selecting the ADD PIN entry. A new form line is created, which can then be filled in; you must specify the pad information for at least the Top and Bottom levels, and also for Inner levels if the board is multilayer.

4.2.2 Deleting a Padstack Definition

You may delete a pad stack definition if it is not required, i.e. not unique. Select DEL PIN and the prompt displayed is:

Select Pin to Delete>

Select the definition to be delete with the cursor. When you do so, the message:

About to Delete Pad Stack for Pin [X] Confirm Y/N>

Select Y<cr> or F1 to confirm. Pins whose padstack definition are changed will take on the definition for All pins.

4.2.3 Adding a Level Definition

Additional levels can be defined for existing padstacks by selecting the ADD LVL command. You will be prompted to select a padstack definition to add a level definition to with the message:

Select Pin to Which New Level is to be Added>

When the padstack is selected with the cursor, another prompt appears, requesting the pin:

Enter new level for pinX>

When you type the level, a new pad stack entry is created, and can be filled in with the appropriate data.

4.2.4 Deleting a Level Definition

Level definitions may be deleted with the DEL LVL command. This may be done if the level's pad definition is no longer required or unique. When the command is selected, the prompt:

Select Level to Delete

is displayed. Pick the level with the cursor, and the entry disappears.

NOTE: Top, Bottom, and in the case of multi-layer boards, Inner level definitions cannot be deleted. If you attempt to do so, the message displayed is:

Top, Bottom, or Inner Layer Levels cannot be deleted.

If the intent is NOT to have a pad on a level, use the entry of 0 Size and R Shape (Round).

4.2.5 Changing a Padstack Definition

You may change existing pad stack definitions by putting the cursor on the entry to be changed and selecting SELECT. Then type the new value into the entry. The units must match the current system unit.

4.2.6 Completing the Padstack Definition

When all entries have been completed, you may save the definition by selecting COMPLETE, not the EXIT command. This first checks the definition for correctness. The items checked are as follows:
• A check is made that there are no pin number conflicts in the entries. Since an entry can be a range of numbers, a pin number can potentially have its padstack defined on different line entries. This an error and must be corrected.
• A check is made that all of the necessary data fields are filled in. In particular, annular and finger pads require additional information.
• A check is made that the data in the entries is valid.

If any errors are found, a message is printed and the entry in error is marked with a flashing cursor. Only the first error is marked, so additional errors may occur when you try to complete after fixing the error. If no errors are found, the Padstack definition command is exited and the Part Decal padstack is updated.

If you wish to exit without saving the changes made, select EXIT (F10).

4.3 Changing the Size of Text and Lines

PADS-2000 gives you the capability of individually changing the line width and text size of almost all items in the database. It is sometimes useful to change all lines of a certain size to a new size. This would be useful, for example, if your photoplotter lacked an aperture corresponding to a selected track width. It is easier to change all occurrences of the line width with a single global command, than to select each track and change its width. This is done either while adding the item to the database or with the Size command.

When selected, the menu changes to Figure 4.4:

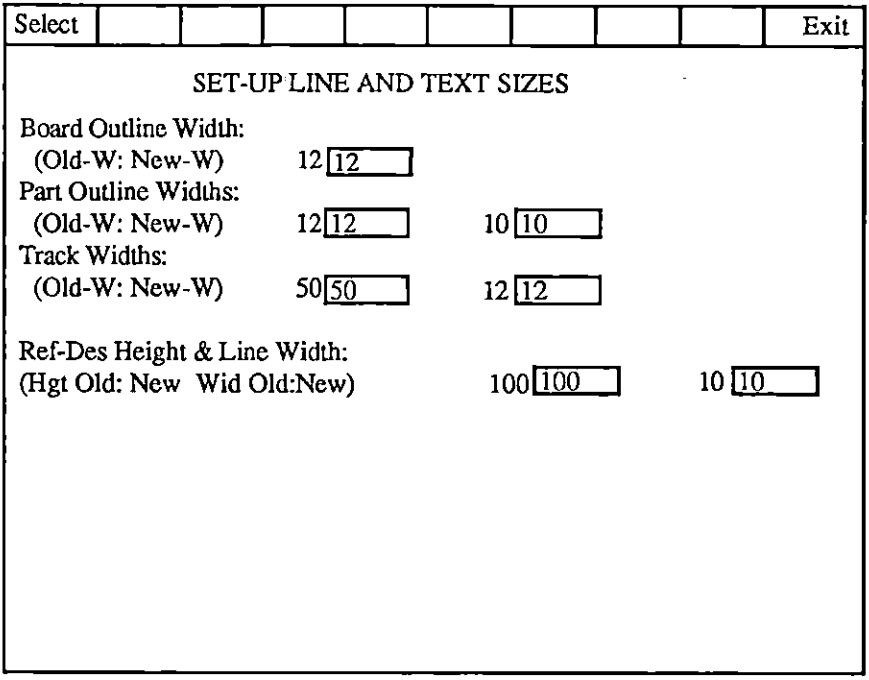

Select
Exit
SET-UP LINE AND TEXT SIZES
Board Outline Width:
(Old-W: New-W) 12 12
Part Outline Widths:
(Old-W: New-W) 12 12 10 10
Track Widths:
(Old-W: New-W) 50 50 12 12
Ref-Des Height & Line Width:
(Hgt Old: New Wid Old:New) 100 100 10 10

Figure 4.4 Text and Lines Form Window

You may change one or more entries. When you exit from the option, all entries that are changed will be changed in the database, and the screen will be redrawn with the new widths.

4.3.1 Changing the Width of the Board Outline

The first entry in the form is the board outline. This is followed by the current value, and an editable entry. To change the width of the board outline, select the editable entry with the mouse. Then, type the new value.

4.3.2 Changing the Width of the 2-D Lines

For each line width used to draw general graphics there will be one entry in the form. To change the width of any of the entries, select the editable entry with the mouse. Then, type the new value at the prompt line. All lines of this width will be changed to the new value.

4.3.3 Changing the Line Widths in Part Decals

The lines used to draw Part Outlines are changeable with the next entries. There will a line for each different line width in one or more Part Decals.

Good library creation practice is to use a minimum number of different line widths, so you should develop a standard that all engineers creating library parts adhere to.

4.3.4 Changing Tracks Globally

Each of the Track Widths used in the design is listed. You may change one or more of the track widths with this command. When a track width is changed, all track segments of this width will be changed. Be sure to run a DRC check after making a global track width change.

4.3.5 Defining the Width of Non-Poured Copper Areas

Copper area drawing apertures may be changed in the Size menu. Changing the copper area aperture has two effects:

1) It changes the area of the copper, because copper areas are drawn with a line of this width on the outline. The copper will overhang the area you create by half of the width of the copper draw aperture.

2) The copper is plotted either as a solid copper area or a cross-hatched area. A small copper size of, say .012" on a grid of .050", will produce a cross-hatched, whereas an aperture size of .050" on a .050" grid will be solid.

4.3.6 Poured Copper

The Poured Copper entry is used to change the width of the aperture used to draw poured copper areas on the board. The copper aperture is used to plot either a solid poured copper area or a cross-hatched area. A small copper size of, say .012" on a grid of .050", will produce a cross-hatched, whereas an aperture size of .050" on a .050" grid will be solid.

4.3.7 Setting the Size of Reference Designators

The Ref Des entry is used to set up the size and line width of the text used to show the Part names and Part Types in the layout (all names are the same size.) To change the text size assigned to part names, select one of the two fields: the Height or the Line (or draw spot used to draw the text character.) Then, type the new value.

4.3.8 Setting the Size of Other Text

The size of all other text items in the database can be changed globally. Each of the different text sizes in the data base is listed. To change the global size for the text item you wish to change, select one of the two fields: the Height or the Width (or draw spot,) used to draw the text character. Then, type the new value.

4.4 Setting System Parameters

PADS-2000 has a number of system parameters that allow the user to control the design environment. These parameters are accessible in a form window displayed when the PARAMS command is selected.

The first four options, are shown with a light blue background. These options can have a value of either On or Off. The value is changed each time the entry's blue box is selected with the mouse.

The remaining entries are changed by selecting with the mouse and typing in the new value, followed by a <CR>.

After you have changed all desired parameters, Exit (F10). The display will be redrawn with the new selections active.

The system parameter defaults are stored in a file called \PADS\STARTUPL.ASC. After you have made changes, the new parameter settings are stored with the job. When the job is brought back into the system, the changed values are as you had changed them.

This section explains the purpose for each of the parameters.

4.4.1 The Show Level Parameter

SHOW CURRENT LEVEL LAST provides the designer with an option for controlling the priority of the level displayed, during the interactive routing of connections. It may be set On or Off.

If the option is Off, items on different levels are displayed in a fixed order that is, Level 1 items on top, then level 2, then level 3, etc. For example: A Pad Stack with pads on Levels 1, 2, 3 and 4 will have Level 1 shown, the other 3 levels are hidden behind the Level 1 pads. If a track on level 4 occupies the same position as a track on Layer 1, it will be hidden under the track on Layer 1.

When the option is On, the data on the current active level will be displayed on top of all other data. This is useful during interactive routing. For example: if routing on Layer 2, the designer may wish the items (pads, vias and tracks) on Layer 2 to be displayed over items on other levels. When the designer goes from another level to Level 2 with the option On, the display will repaint with Level 2 items on the top of all other layers. That is to say, the pads, vias and tracks on Layer 2 are shown, and the pads, vias and concurrently located tracks will be behind and hidden behind those on Layer 2.

Every time a layer change is made, the screen will be redrawn with data on the new current layer displayed over the other layers.

NOTE: The SHOW LEVEL command may be also turned ON and OFF, using the Modeless Command of D <cr>.

4.4.2 Highlight

The CURRENT NET HIGHLIGHT option enables the user to switch on or off the Highlight net mode. This option has two settings, On or Off. Highlight means that the color of the net including PADS will change to the other color of the "color pair" currently being displayed. All colors in the palette are paired; these pairs are next to each other in the color palette of the display option. For example, if the tracks are being displayed as Dark Red, the highlighted color of the selected Net will be a Light Red. When the option is set On, the entire net that is currently being routed or edited is highlighted. When set to Off, only the route is highlighted.

We recommend that you normally set tracks on various layers with the darker of the two colors in a color pair if you intend to use the highlight command. This will allow highlighted colors to stand out.

NOTE: The HIGHLIGHT command may be also turned ON and OFF, using the Modeless Command of H <CR>.

4.4.3 Displaying Tear Drops in the Design

It is generally felt that widening tracks at the point where they join pads and vias will lead to higher PCB manufacturing yields and circuit reliability. There is less chance of copper smearing or drill-hole breakaway. This track widening effect is called a "Tear Drop", and is supported in PADS-2000.

The TEAR DROP PAD GENERATION entry has two options, On and Off. When switched On, tear drops are added to every track where it joins a pad, based on the following rules:

1) Every track that connects to a round or annular is converted to a tear drop. Every track that connects to the end of an oval finger pad will be converted to a tear drop. Tear drops will not be added to tracks that connect to square , rectangular, or odd shaped pads.

2) The size of the tear drop is a function of the track size and pad size, and it is oriented in the direction that the track enters the pad.

3) If the track segment entering the pad is not sufficiently long to allow a tear drop to be added, or if the track enters the pad "off angle" or not directly at its center, no tear drop will be added.

4) If the track is greater than 2/3 of the pad diameter, no tear drop will be added.

All tracks in the database will be "tear-dropped". You have the option to turn off (deselect) specific tear drops in the Route menu.

NOTE: Tear drops may in some cases lead to DRC rules violations, therefore, always run the DRC program after setting teardrops.

NOTE: Tear drops are considered track segments, so obey the rules for track segments set in display, track width, etc.

NOTE: Tear drops will be output by the CAM program to all plotting devices. For standard line drawing penplotters and photoplotters, adding tear drops will increase total plotting time and redraw time.

4.4.4 Setting the Display Mode for Poured Copper

The COPPER POUR HATCH DISPLAY parameter allows the user to select whether to show Poured copper areas with just the outline or with the actual copper hatch area drawn including all cutouts for pads and tracks. The parameter has two options, On or Off.

When set to On, all copper that has been created with the Copper Pour command will be drawn showing the copper hatch fill pattern exactly as it will appear on the artwork output. This includes cutouts, and spacing around tracks and pads.

When the option is set Off, only the outline of the copper area is shown, along with the outline of the cutout area.

The advantage of showing only outlines is that graphics drawing speed is faster. The advantage of showing the copper hatch is its displayed design more accurately reflects the final PCB artwork.

4.4.5 Setting the Displayed Dot Grid.

The Dot Grid parameter is used to change the spacings between the Dot Grid displayed at the CRT. The Dot Grid is a visible guide for drawing lines, decals, placing pads, etc. It is independent of the System Grid (or snap grid). The dot grid is homogeneous in X and Y dimensions. Type in the value that you want for the spacing between dots.

If the grid is not visible, it is because the distance between dot grids is small and the display is zoomed out so far the number of dots would be enormous and not meaningful. Change the dot grid or zoom in and the dots will appear.

4.4.6 Setting Tracks and Lines to Real Width

The purpose of REAL WIDTH is to allow tracks to be displayed at their "real width" , i.e..012" wide, .050" wide, etc. Tracks lines, and text with a width less than the selected width will not have True Width displayed. They will be displayed with their center line only .

The rationale behind allowing the designer to select the widths to be displayed at their real width is to permit the user to "trade-off" between display redraw time and realism of the displayed information. Repaint of the display, using track center line display only is faster than when using the true width, but it is also considerably slower.

NOTE: The REAL WIDTH selection, in addition to controlling the width of the track displayed at its real width, also controls the width of text, 2-D lines and decal outlines, i.e., the width selected for tracks also applies to these three additional items.

NOTE: The Real Width can also be set using the modeless command R.

4.4.7 Setting Thermal Line Width

This parameter defines the line width used to draw thermal relief lines which tie to the copper pour plane. Changing this value after a copper area has been flooded requires the selection of FLOOD again.

4.4.8 Setting the Hatch Grid for Copper

This parameter defines the spacing between the lines used to draw the Poured Copper. Poured Copper is drawn with a cross hatch of a user defined separation. You set the spacing for all Poured copper areas with this command.

The recommended Poured Copper hatch grid parameter setting should be either .025" or .050". If the copper is drawn with an aperture of .050", this will create solid copper. If the copper draw aperture is less than .050", the copper will be drawn with a hatch pattern.

NOTE: Do not use a Draw aperture of less than .025" unless you have an abundance of RAM memory.

4.4.9 Copper Pour Clearance Parameter

Poured Copper will automatically create a clearance area around any pad or track that is not part of the signal for the copper. You may define the size of the clearance with this parameter.

4.4.10 Design Rule Checking Clearance Parameters

The three DRC parameters, Pad to Pad, Pad to Track, and Track to Track, enable the user to enter the spacing rules between various conductive surfaces on the board. This entry is used for reference during the execution of the major menu Checking.

NOTE: Copper and Text on a signal layer are also checked for design rule spacing, against pads, tracks, vias, etc. For space checking, copper and text are treated as if they are tracks.

4.4.11 Setting the Interval Between Backups

This parameter lets you define the interval between automatic backups. PADS-2000 automatically makes a Backup file of the database in memory at set intervals of time. The purpose of the Backup file is to give you a convenient way of recovering your work investment, in the event of an emergency such as a power failure, hardware failure, a

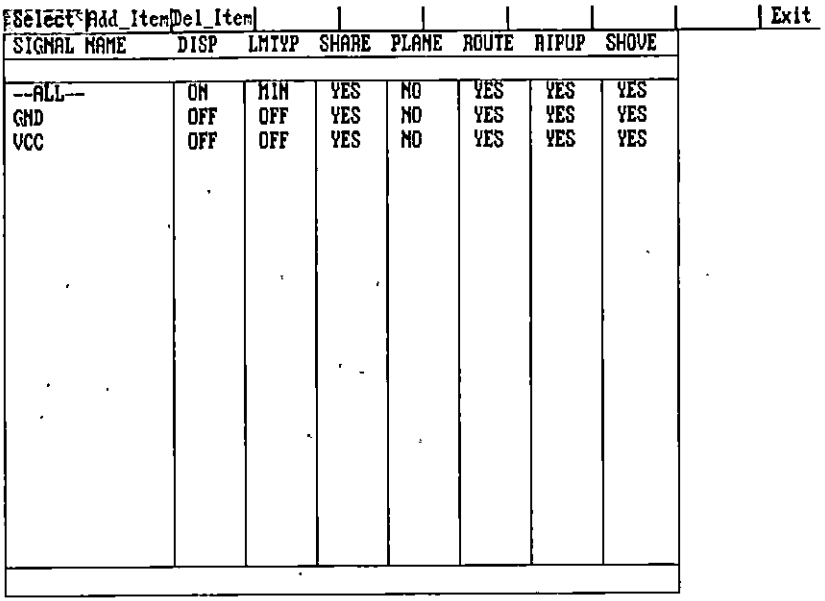


Figure 4.5 - NET ATTR Menu

4.5.1 Selecting the Nets to Display

In the first column are listed the names of the nets that are to be controlled. Any net name not listed will be considered part of the All group. To add or delete nets from the Net Control Window, use 2. ADD ITEM or 3. DEL ITEM, as applicable. When a new net is added to the list, it takes on as its default parameter settings the current setting for ALL connections.

Net names can be added individually, or buses can be added with a wildcard extension, e.g. DATA1, DATA2, and DATA3 can be added by typing:

DATA*.

All of the nets in the database that start with DATA will be displayed in the menu. If the number of nets exceeds the lines in the window, the

last selection in the window will be MORE. To view additional nets, select MORE with the left mouse button. This acts in a fashion similar to the Page Down key, to select the next page of the Net Display.

4.5.2 Controlling the Display of Nets

The second column, DISP, lists for each net the rules to be used in displaying its tracks and connections. The four choices are: On, Off, PR1, and PR2. These control the display of tracks and connections as follows:

- When a net display is On, all tracks and connections in the net will be displayed.
- When the net display is Off, no tracks or connections in the net will be displayed. Also, neither the Autorouter nor Length Min commands will operate on the net.
- When the net display is set to PR1 (Partial 1), all completed tracks and partial tracks will be displayed, unrouted connections of partial track are not displayed, however, all remaining unrouted connections are displayed.
- When the net display is set to PR2 (Partial 2), all completed tracks and partial tracks are displayed; all connections are not displayed.

NOTE: PR1 and PR2 are to be used only on PWR and GRD Nets when PWR and GND planes are used. In this instance, P1 and P2 can be used as a visual aid in creating "Exit Vias" to the plane.

4.5.3 Selecting the Length Minimization Rules

The next column, LMTYPE lists for each net the rules to be used in establishing the rats nest connection pattern. There are four condition states for each net or net category in the window. These are: Off, Horz, Vert, MinD and ECL. A state condition is selected by placing the cursor over the connection type and selecting" until the desired state condition is obtained.

MinD reorders the connections within a net so that each pin is connected to its nearest neighbor(s),and the total length of the net is the minimum possible.

A "before" and "after" situation of MinD is shown in Figure 4.6.

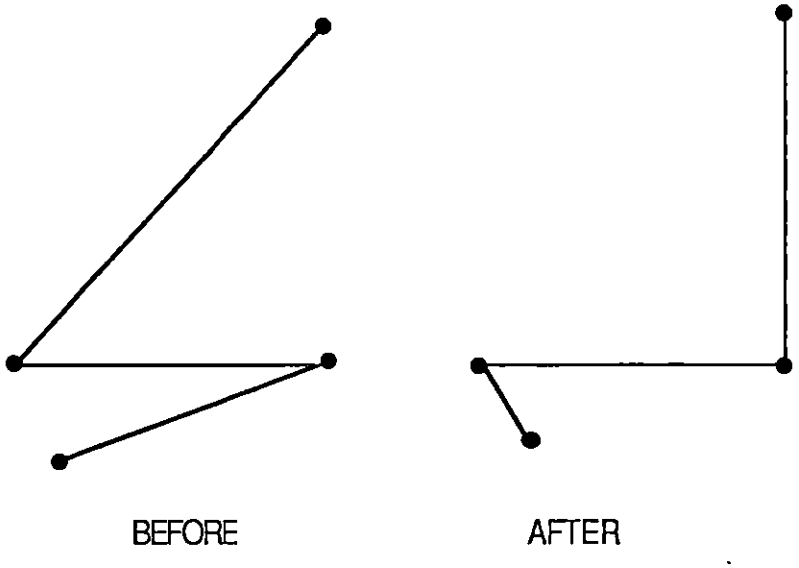


Figure 4.6 - Effect of Length Min on MinD Net

VERT and HORI will force the connection pattern to be predominantly vertical or horizontal. A minimum connection length calculation is run in an attempt to find the minimum pattern. However, connections that go in the direction opposite of the bias selected will be weighted by a factor of 10. For example, if VERT were selected, the software would prefer to put in a vertical connection 10" long to make a connection rather than one 1" long that is horizontal. The resulting pattern is very good for power and ground bus connections.

Selecting ECL will result in the connection not being changed from the original order input from the Logic Capture system.

Selecting the NO minimization option for a net will prohibit any length minimization from occurring. This option should be limited for use in high-speed circuitry, where the ordering of the net is critical to its function.

4.5.4 Setting Copper Share Rules

The Share column is used to enable or prohibit copper sharing by the auto-interactive router. There are two options, ON to allow copper sharing and OFF to prohibit it. It should be On for all tracks that do not require high-speed "daisy chain" style routing.

4.5.5 Defining the Power Plane Layers

The next column, PLANE, defines which signals, if any, are to appear on plane layers. This information is read by the optional PADS-SuperRouter autorouter when routing power tracks from SMT parts. Routes from an SMT part to a power plane will automatically end with a via in the Superrouter if the signal is defined as a plane layer. This via is used to tie the pad into the plane. This information is also used by the Check Plane command, see section 8.6 for details.

4.5.6 Defining the Nets to be Routed by PADS-SuperRouter

The column headed ROUTE defines which nets in the database are to be routed by PADS-SuperRouter. If a net is marked No, it will not be routed; if marked Yes, it will be routed by PADS-SuperRouter.

4.5.7 Setting the Rip Up Rules for Nets

The Ripup column is used to enable or prohibit track rip-up by the auto-interactive router and by PADS-SuperRouter. There are two options: ON to allow ripup and OFF to prohibit it. If set to Off, the track is fixed in position and will not be moved even if this would improve the routing results. If set to On, the router may move the track if this will improve the routing. Unless you wish to fix a track exactly in position, this option should be set On.

4.5.8 Setting the Shove Rules for Nets

The next column, Shove, enables the user to allow or prohibit a net from being moved by the Pushing algorithm in the Route menu. If set to Off, the track is fixed in position and will not be moved even if this

would improve the routing results. If set to On, the router may move the track if this will improve the routing. Unless you wish to fix a track exactly in position, this option should be set On.

4.6 The Macro Command Language

A Macro is a collection of keyboard and mouse events that can be replayed immediately (Auto Macro) or at a later time (User Macro), and in the process, save a considerable amount of the Designer's time, since one Macro can contain many, many individual commands. In short, Macros are a form of Command Shorthand.

A Macro is replayed and performs its intended function when it's name is typed. A macro can be named using any one of the following methods:

1. ALT A through ALT Z (case insensitive)
2. ALT F1 through F10
3. SHIFT F1 through F10
4. CONTROL F1 through F10

All macros are stored in memory for quick replay. The number of macros that can be stored is limited by the number of names available (currently 56) and the amount of memory available (actual limits are discussed later). The total number of keystrokes for all macros that can be stored is limited by available memory. There is no limit on the number of keystrokes for an individual macro other than the memory limit.

The Macro Management Menu is found in SETUP, F3 MACROS. When selected, the menu changes to the following:

1 Status 2 LoadFile 3 Save 4 Del Mac 10 Exit

4.6.1 Creating a Macro

Macros are created one of two ways: from within PADS-2000 "By Example" or by using a Text Editor and writing a macro file that will be loaded later at run time. Both forms of creation and the format of a macro file are discussed later.

4.6.2 Display Resolution Independence and Coordinates Within Macros

Whenever anything on the screen is picked during macro creation, whether it is a component or a choice in a pop-up menu, a coordinate is stored in the macro. If that macro is played back with PADS-2000 configured for a graphics adapter with a different resolution, the stored coordinates must point to where the user originally intended. This is referred to as resolution independence. Even though the above situation probably won't happen that often, it's good to know a little something about it. Picking objects on the circuit board, i.e., components, routes, etc., doesn't pose much of a problem for resolution independence as those "database" coordinates are by definition, already independent of display resolution. The ones to worry about are picks in pop-up menus and forms. The macro system tries to store these coordinates in as resolution independent a fashion as possible. The user can help by picking these objects as close to the object as possible.

4.6.3 Absolute Versus Relative Coordinates

Macros can have two kinds of coordinates: Absolute and Relative. An Absolute coordinate is independent of all other coordinates in that macro. A relative coordinate is always related to its immediate predecessor (i.e., move 100 units in the X and -50 units in the Y from the previous coordinate.)

The User Macro always defaults to the "ABSOLUTE" coordinate mode. A macro with Absolute coordinates, when replayed, always generates the same coordinates with which it was created. A good example is a macro that fills a form or picks a choice from a pop-up. Forms and pop-ups always appear at the same place on the screen and, therefore, the macro should always replay at that place.

A macro with relative coordinates, when replayed, generates coordinates that are relative to the cursor position at the time replay starts. An example is a macro that does some repetitive, step and repeat function, such as connecting a memory to an address bus, or picking an array of parts of the schematic. The user would create a macro that adds one terminal (using relative coordinates). The cursor is then moved to the

next position for a terminal to be added and the macro is replayed. Since all the coordinates in this macro are relative to the cursor, this terminal will be connected in the same fashion as the first.

4.6.4 The Auto Macro

There is a special kind of macro called the Auto Macro. It does not need a name and hence there is only one Auto Macro at any given time. (The other type of macros, the ones with names, are called "User" macros.) The Auto macro exists mainly as a quick way to make a temporary macro that you intend to use only once and then discard. This macro cannot be saved to a file.

4.6.5 Loading a Macro File

Macro files can be loaded two ways. One way is to put the macros that you always want loaded in the file PCB.MAC in the PADS\FILES directory. This file is always automatically loaded at program startup. The second way is to load from within PADS-2000 by using the LOAD Command on the MACRO menu. The LOAD command will always load all the macros in a file. If any of the macros in the file already exist in memory, the load process will be aborted.

4.6.6 Macro Replay Control

There are two types of control you have while a macro is replaying: Pausing and Aborting. To pause during replay, use the Space bar. To resume, use the Space bar again. To abort replay, type ^X (Control-X).

4.6.7 Yes/No Questions

There are many places in the software where a Yes/No question needs to be answered while creating a Macro. These include various phases of Part Editing as well as when Exiting the system. Not all of these questions are asked every time. For instance, if you try to save over an existing part or file, you will be asked to verify. When creating a macro you need a way to deal with this situation.

The way to control this is with the Yes/No mode. This mode tells the macro system how to deal with Yes/No questions. The Yes/No mode affects both the way macros are created and the way they are replayed.

For instance, if the mode is anything other than M (Macro) and a Yes/No question is encountered during macro creation, the answer to that Yes/No question is not stored in the macro. During replay, if the mode is anything other than A (Ask), the question is answered either by the system or from the macro. There are several modes--each is discussed below.

mode = Y (yes) During replay, always answer YES. During creation, don't store the answer to any Yes/No questions.

mode = N (No) During replay, always answer NO. During creation, don't store the answer to any Yes/No questions.

mode = A (Ask) During replay, if a Yes/No question occurs, the user is asked for the answer. During creation, don't store the answer to any Yes/No questions.

mode = M (Macro) During replay, if a Yes/No question occurs, get the answer from the macro. During creation, STORE the answer to any Yes/No questions.

mode = R (Restore) (see below)

4.6.8 Yes/No Restore Mode

This mode provides a way for a macro to temporarily change the Yes/No mode and then restore it to what it was previously. If a (Y/N) command is encountered during replay, the old mode is remembered by the macro system. When the (Y/N) command (Set Yes/No mode, restore) is encountered, the mode is restored.

4.6.9 Macros Can Call Other Macros

If a macro is invoked during macro creation, the call to that macro is stored in the one being created.

4.6.10 Macro Commands

The following is a summary of the macro related commands. Following the summary, some of the commands are discussed in detail.

* alt 1 = replay auto macro
* alt 2 = create/save auto macro
* alt 3 = create/save user macro
* alt 4 = set coordinate mode
* alt 6 = set yes/no mode
* alt 7 = restore position
* alt 8 = go to top menu
* alt 9 = display file
* alt 0 = display errors
* C = set macro replay count
* F1 = Status
* F2 = Load file
* F3 = Save macros to file
* F4 = Delete macro

4.6.11 Create Auto/User Macro

These commands are toggles, hit it once to begin creating a macro, hit it again when the macro is complete. When creating a user macro, you will be asked for the macro name (the key used to invoke replay of the macro) and an optional description. When user macro creation is complete, you will receive a message to that effect; including the current status of the macro system. The status tells you how many macros are defined, their total length (in keystrokes) and the percentage of available macro memory in use.

When you complete creation of a macro, the current cursor position is stored in the macro as a MOVE command.

4.6.12 Set Absolute/Relative Coordinate Mode

This command toggles the mode between Absolute and Relative. As stated above, CREATE AUTO MACRO always starts in Relative mode. CREATE USER MACRO defaults to the Absolute mode.

The following is a VERY IMPORTANT NOTE: When creating a User Macro, using Relative coordinates, make sure the coordinate mode is toggled from Absolute to Relative before the first coordinate is stored. This initializes the relative coordinate system and ensures that the first relative coordinate is relative to the cursor.

4.6.13 Restore Menu Position

Whenever a macro is invoked, the current menu position is stored. The restore command, when encounted during playback, will attempt to place the user at that saved position. When encountered during creation, this command will write a restore command to the macro (it will not do a restore).

There are certain situations in which the restore command will not restore to completion. These include inconsistent Yes/No questions and menu commands that require the user to type a response (such as entering a Decal Name to MOD DECAL).

4.6.14 Top Menu

This command will attempt to move to the top most menu. The operation of this command is tied closely to that of the Yes/No command since many Yes/No questions may be encountered on the way to the top. If this command is encountered during macro creation, a Top command is written to the macro and then a Top command is performed.

4.6.15 Display File

While this command is not part of the macro system, this is where you'll find out about it. This command displays a file in the pop-up window. Lines that are too long are wrapped around. Files that are too

big are truncated. If no path is given in the filename, the PADS\FILES directory is assumed. While the file is displayed, the following commands are available:

Home	Display first screen
End	Display last screen
PgUp	Display previous screen
PgDn	Display next screen
Esc	Exit file display

4.6.16 Display Errors

This is a special version of the Display File command. If error messages are being redirected to file PCB.ERR, this command will display that file. If errors are going to the printer, you'll receive a message to that effect.

4.6.17 Set Macro Replay Count

This command specifies the number of times a macro will be replayed when it is invoked. It is primarily for use with auto macros but can be used with user macros. To replay a macro only once, just type the macro name. To replay it more than once, type "c" followed by the number of times you want it to replay (no <CR>), then type the macro name. For instance, to replay the alt-a macro 3 times, type c3 and then alt a.

This command only affects the current macro replay. The next macro to replay (in the absence of a "c" command) will replay only once.

The "c" command can be stored in a macro. Therefore, a macro A can call macro B 3 times.

4.6.18 Help/Status

This command displays in a pop-up window (using the pop-up window commands from above) some macro help text as well as the current status of the macro system. This status includes one line for each defined macro, each line containing the following:

* the macro name
* the length of the macro in keystrokes
* the percentage of macro memory the macro occupies
* the macro description, if any

The last line gives the total length of all macros and the total percentage of macro memory used.

4.6.19 Load File

This command is used to load a macro file to memory. If no path is given in the filename, the PADS\FILES directory is assumed.

4.6.20 Save File

This command saves (writes) ALL the currently defined macros to the specified file. If no path is given in the filename, the PADS\FILES directory is assumed. This command will OVERWRITE any existing file of the same name.

4.6.21 Delete a Macro

This command deletes one or all macros from memory. (It does not affect macro files.) To delete a single macro, enter the macro name. To delete all macros, enter "*". Take care with the use of *. It will delete ALL your Macros.

4.6.22 Defaults

Defaults at Program Startup:
Coordinate mode is Absolute
Yes/No mode is A (ask)
Macro replay count is 1.

4.6.23 Limits

Total number of keystrokes	4096
Total number of macros	56
Maximum nesting depth	64

4.6.24 Macro Language

The proceeding portion of this chapter described the creation of Macros using the cursor, the Keyboard and Menu Commands to create the Macros and record it while actually doing the intended Macro.

In some instances, it is better to use a Text Editor (PEDIT, WORD PERFECT, etc.) to create an ASCII file of the Macro. To do this requires a knowledge of the Macro language.

The following describes the elements of the macro language. Unless otherwise specified, all elements are upper/lower case insensitive.

Comment
(*XXX*) where XXX is any number of characters as long as the total including the "(*" and "*)" does not exceed 80 and everything is on one line.

F-Keys
(F1)
(F2)
(F3)
(F4)
(F5)
(F6)
(F7)
(F8)
(F9)
(F10)
F11 and F12 are not supported

Numeric Keypad Keys
(home)
(end)
(up)

(down)
(left)
(right)
(pgup)
(pgdn)
(ins)
(del)

Control Characters

If a macro must generate control characters such as Carriage Return or Tab, the following must be used.

Macro Element	Control Character	Description
(soh)	^A	
(stx)	^B	
(etx)	^C	
(eot)	^D	
(enq)	^E	
(ack))	^F	
(bel)	^G	
(bs)	^H	Backspace
(ht)	^I	Tab
(lf)	^J	Linefeed
(vt)	^K	
(ff)	^L	Formfeed
(cr)	^M	Carriage Return (Enter)
(so)	^N	
(si)	^O	
(dle)	^P	
(dc1)	^Q	
(dc2)	^R	
(dc3)	^S	
(dc4)	^T	
(nak)	^U	
(syn)	^V	
(etb)	^W	
(can)	^X	
(em)	^Y	
(sub)	^Z	
(esc)	^[	Escape

(fs)	^\	
(gs)	^]	
(rs)	^^	
(us)	^_	
(btab)	none	Back Tab

Alt, Control and Shift

Any of the above can be Alt'd, Ctrl'l or Shifted (where appropriate) with the addition of a-, c- or s- respectively. For instance Alt-F1 is (a-f1), Ctrl-F1 is (s-f1), and Shift -F1 is (s-f1).

Macro Statements

(bdef) (*desc*) (key)	begin macro definition
(edef)	end macro definition
(rstr)	return to saved menu location
(top)	return to first menu
(yn) n	set Yes/No mode (y(es), n(o), a (sk), m(acro), r(eset))
(move) x y	move cursor to (x,y)
(rel)	coordinate following next command is relative

4.6.25 Sample Macros

A Macro to Unconditionally Exit the System

(bdef) (*exit*) (a-x)	(*begin definition of macro alt-x*)
(yn) y	(*answer Yes to all Yes/No*)
(top)	(*go to the top*)
(f10)	(*exit*)
(edef)	(*end of definition*)

A Macro to Get to the Macro Management Menu

(bdef) (*exit*) (a-m)	(*begin definition of macro alt-m*)
(yn) a	(*ask me at yes/no*/)
(top)	(*go to the top*)
(f2)	(*Setup*)
(f6)	(*Macro*)
(edef)	(*end of definition*)

A Macro to Save a Specific PADS-2000 File

```
(bdef) (*analog*) (s-f1)   (*begin defn of macro shift-F1*)
(top)                      (*go to the top*)
(f1)                       (*In/Out*)
(yn) y                     (*answer yes at yes/no*)
(f2)                       (*Job Out*)
analog (cr)                (*name of job file*)
(yn) r                     (*reset yes/no mode*)
(rstr)                     (*restore menu position*)
```

4.6.26 Modeless Commands

A somewhat different form of Macros are the Modeless Macros or Modeless Commands provided by CAD Software and contained in the software.

Although strictly speaking, these are not true Macros, since their use is to save key strokes and thus time, they are contained within this chapter on Macros.

CAD Software has built into PADS-2000, a number of "time savers" that you should use. Generally, these are single key strokes programmed to perform a specific function.

The following lists these Modeless Commands and their function. They are all case insensitive. Become familiar with them.

WYY — Changes current Width to XX mils.

GZ — Changes the current System Grid to Z mils

SU1 — Searches for Part U1 and centers the cursor on the Origin of the Part.

ALT0 through ALT9: — Reserved for Macro Command

Cn — The following macro will be replayed n times.

NOTE: Modeless Macros function at the higher levels of the Menu structure, but depending upon the Modeless Macro involved, may not

function at a lower level. Example: Search (SU1) will NOT function if in the MOVE routine.

4.7 Setting the System Units

The UNITS command lets you select the dimension unit for PADS-2000 to be either inch or metric. There are three choices: Metric, Inch, or Thousandths of an Inch,. When selected, respond to the prompt:

Units are Metric, Mils, or Inches [Mils] (ME/MI/I)>

Type:

ME<CR>

for Metric,

I<CR>

for Inch,

MI<CR>

for Mil or 1 Thousandths of an inch.

When Metric is selected, all of the display values convert to metric. The basic unit is millimeters and the possible resolution is .001 mm. Your input will be in millimeters, for example 1.345 mm is input as 1.345 <CR>.

When Inch is selected, the unit is inches, and the resolution is .00001". Your input will be in inches, for example 1.645" is input as 1.645 <CR>.

When Mils is selected, the unit is .001", and the resolution is .00001". Your input will be in .001", for example 1.645" is input as 1645 <CR>.

You must remember to enter all dimensional values requested by the system using the current coordinate system. You are free to move between metric inch and thousandths of an inch coordinate systems at any time in the design. The fundamental system grid insures that there is no loss of data accuracy when you switch coordinate systems.

NOTE: When items are stored in a file, the database is stored in basic database unit, regardless of whether the entry was in Inch or Metric. This does not affect the display of data.

4.8 Establishing the X,Y Origin of the Design

The Origin command allows you to move the System 0,0 origin to a location within the 56" x 56" design database.

Establishing a new system origin is done by first placing the mouse cursor at the desired location and then selecting the Origin command. The Origin symbol will appear at the cursor location and the message:

NEW ORIGIN IS AT SYSTEM LOCATION (XXXX,YYYY)

appears. When this New origin has been so located, the entire coordinate reference system is no longer referenced to the original System 0, 0 but to the new origin location.

After a new origin location has been selected, the X/Y cursor location, positional information is, with respect to the New origin location, not the original Origin 0, 0 location.

This command is useful mainly during the production of CAM outputs. Many CAM machines require that the system origin point for all data be referenced to a specific tooling hole or feature in the board. This command lets you reset the location at any time.

The command is also useful when parts are being added to the layout either via an input ASCII command or ECO In command. Parts are added at the system origin, so placing it a short distance outside the board outline can be very useful.

NOTE 1: Part Decals have their associated Local Origin built into the Part Decal and it automatically appears with the Part Name and Part Type when Create Part Decal is used.

4.9 Setting the Parameters for PADS-SuperRouter

The SR PARAMS command is used to control certain parameters during routing with PADS-SuperRouter. The features that can be controlled by the Setup File are as follows:

- The Direction of Routing on each Layer

5.3.4.1 Moving a Corner

The MOVE CRN (F1) command is used to move a corner of the Board Outline. When a corner is moved, the two segments that intersect to make the corner are also moved. There are two options available for how the connecting segments are treated as you move the corner: Orthogonal or Angle. If you are moving the corners Orthogonally, the attached segments will maintain their previous angle of intersection. This will require that the other ends of the segments move as you move the corner. If you are moving in Angle mode, the two intersecting line segments forming the corner will change the angle of intersection as the corner is moved, and the other end of these line segments will remain fixed.

Place the cursor over the desired corner and select MOVE CRN. The corner will be picked and attached to the cursor, and will be free to move as you move the cursor. Note that the sides of the board that are joined at the corner also move.

The default mode for moving is Orthogonal. You can toggle the Angle mode on/off by selecting ANGLE (F2). The mode of operation changes at the exact location when you change the mode.

When you are satisfied with the location of the corner, select COMPLETE (F1) to set it.

EXIT (F10) will cancel the Move Corner option, returning the board outline to its original position.

5.3.4.2 Moving a Segment

This command is used to move a segment of the Board Outline. As the segment moves, the segments that are attached to it will remain attached and they will be lengthened or shortened, as required. Segments may be moved only horizontally or vertically.

To use, place the cursor over the desired segment and select MOVE SEG (F2). The segment will be attached to the cursor and can be moved in either the X or the Y direction. With the segment attached to

the cursor, move the cursor to place the line segment at the new location.

COMPLETE (F1) is used to place the segment in the new location.

EXIT (F10) will cancel the command, returning the segment to its original location.

5.3.4.3 Cutting Segments in a Line

The CUT SEG command allows the designer to break a line into two different segments and manipulate each segment in an orthogonal or an angled direction. CUT SEG can be used to cut an orthogonal or diagonal line.

When an orthogonal line is selected, the line is broken into two at the point of the cut. These two lines are joined by a third new line perpendicular to the first two. One segment remains in place and the other segment can be moved. Moving the cursor will move the "movable" segment.

If the movable segment is not the segment you wish to move, select SWAP CRN. The original movable section will become fixed and the other segment will become movable.

To move the segment in an angled fashion from a vertical or horizontal, select ANGLE. Both ends of the fixed segment are permanently fixed, and the remaining segments operate in a fashion similar to the Angled mode of Move Corner.

The Cut Segment command will operate in a somewhat different manner when a non-orthogonal line is selected, the line is broken into two non-orthogonal at the point of the cut. These two lines are joined by a third new line perpendicular to the main axis of the original non-orthogonal line.

The new connecting line is constrained to move in X and Y, and the other end points of the two orthogonal lines are fixed. If Angle is selected, one of the lines will be fixed and the other will move.

Select F10 to Exit without retaining the results, or COMPLETE (F1) to set the location of the Cut Segment.

5.3.4.4 Adding Additional Corners to a Board

The ADD CRN (F4) option is used to add a corner to a line in a Board Outline. To use, move the cursor over the line where the corner is to be inserted and select ADD CRN. When the line is picked with the cursor, a corner is inserted in the line and follows the cursor as it moves. The segments that are created from the original will follow the cursor.

Move the corner to the desired location and place it with COMPLETE (F1).

Select EXIT (F10) to cancel the command.

5.3.4.5 Deleting a Corner in the Board Outline

The option DEL CRN is used to delete a corner of the board. To use, place the cursor over the corner to be deleted and select DEL CRN. The corner will be deleted. To the message:

CONFIRM CORNER DELETION Y/(N)?

confirm the deletion of the corner by typing:

Y

The corner will be removed and the previous two segments that intersected will become a single line segment.

When a polygon has been reduced to three corners and an attempt is made to delete one of the three corners, the action will not be accepted since a polygon must have a minimum of three corners.

5.3.4.6 Changing the Line Width for the Board Outline

If you wish to change the width of the line used to draw the board outline, place the cursor over the Board Outline and select LINE WIDTH. The following message appears:

CHANGE WIDTH [12] TO >

indicating .012" is the current Line Width used on the Board Outline. Enter the new Line Width followed by <CR>.

5.3.4.7 Adding Arcs in a Board Outline

A line segment in the polygon may be converted to an arc with the command MOVE ARC (F5). The segment will be converted into an arc whose circumference is bounded by the end points of the arc. As the cursor is moved, the circumference expands to follow the cursor. A read out gives the total number of degrees of the circle that are part of the arc. This reading will be between 0 and 360.

Corners in arced segments can be moved, either by the Move Corner or Move Segment command. Arcs can be moved with the MOVE ARC command. Arced Segments may be changed back to straight segments with the MOVE ARC command.

5.3.5 Moving the Board Outline

The Board Move command is used to move a Board Outline from one location in the work area to a different location and to rotate the board outline.

To use the Move command, place the cursor over the Board Outline and select MOVE. The board will be picked (attached to the cursor), and the menu window will change as shown in Figure 5.7 below:

1 Complete	**2 Rot 90**	**3 Set Ang**	**4 Spin**	**5 Window**
6 Expand	**7 Contract**	**8**	**9**	**10 Exit**

Figure 5.7 - CREATE BOARD, MOVE OPTIONS

Move the board with the cursor; it may be rotated in 90 degree increments, at a set angle, or spinned about its origin if desired. Note that none of the design data is moved with this command, as it applies only to the board outline.

5.3.6 Deleting a Board Outline

Only one board outline may appear in a design database. If you wish to use a different board, you must first delete the current outline. The Delete command is used to delete the outline of the board from the database. Place the cursor over the board outline and select DELETE. To the message:

CONFIRM DELETION Y/(N)?

respond:

Y

5.3.7 Saving the Board in the Library

It is possible to name the board and save it in the Library for use on other designs. Save is used to store the Board Outline in the parts library. Select SAVE. To the message:

SAVE ITEM WITH NAME>

respond with the Name which you want to save the board under. The board will be saved in the Parts Library database (see Chapter 12).

For use with future boards, recall the outline with the Add option described earlier.

NOTE: If it is desired to save the Board as a Job file, then use the IN/OUT Menu. If the board outline contains text always store in the IN/OUT menu since the SAVE option of CREATE does not store text, only the board outline.

5.4 Copper Areas (Both Fixed and Poured)

There are two types of copper in PADS-2000: Poured Copper and Copper. Poured Copper should be used wherever you want PADS-2000 to create isolation areas for pads and tracks that are in the copper area, but not part of the copper's net. The primary use of Poured Copper is to create local ground plane areas on an artwork layer. It should not be used to create autorouter obstacles.

Copper, also known as Fixed Copper, is a solid shape. Pads and tracks that are not part of the signal name for the Copper area will cause shorts. The primary purpose for fixed Copper is to be a heat sink under power components and to act as a routing obstacle to PADS-2000's standard and optional autorouters. It should be used with caution to create ground planes.

Both types of copper can be part of a named signal net. They can both be plotted as a solid object or hatched (only Poured Copper will be displayed as hatched). Poured Copper must be a polygon made up of non-overlapping lines. Fixed Copper does not need to be a polygon, it can also be a line. When it is a polygon, it must be made up of non-overlapping lines. Any number of copper areas or lines can be created or added to the board, as long as the limits for Copper are not exceeded. Copper can be placed only on an Electrical Level or Layer defined by Max Routing Layers in the SetUp Menu.

This section describes the commands for creating, modifying, and removing Copper areas and Poured Copper areas. Most commands are common to both types of copper. Where a command is active for only one type, it is noted.

Poured Copper areas can ONLY be modified with the Copper Pour commands. Similarly, Fixed Copper areas can ONLY be modified with the Copper commands. To distinguish the areas in a design, Pour Copper is shown either with only the outline or Hatched. Fixed Copper is always displayed as a solid area. A Copper line will be drawn at its correct thickness if the Real Width command is set to allow this.

Copper polygons are photo plotted on the Artwork as solid areas or a hatched screen. The choice depends upon the relationship between the line width of the copper used to draw the polygon and the spacing

between the lines of copper when used to fill the copper area during plotting.

During plotting of Artwork, the plotter will draw an outline enclosing the copper polygon using the copper line width centered on the sides of the polygon. The plotter will then draw a series of lines using the copper line width in the vertical and horizontal direction within the polygon, spaced exactly 0.050" apart (This is a default setting in the CAM Menu which can be changed by the designer).

If the area is Poured Copper, the drawn hatch grid will "back off" to avoid unconnected pads and tracks. Circles will be drawn around every unconnected pad. Lines will be drawn around unconnected lines, and thermal reliefs will connect pads that are part of the net.

If the Copper Hatch Grid is .050", then a copper line width selection of .050" will result in a solid filled copper polygon. A copper line width of less than 50 would result in a series of voids, the size of which are determined by the line width. A line width of 0.025" is a good choice for a non-solid copper polygon.

NOTE: If the copper polygon areas on a layer exceed 1-2 square inches, non-solid copper polygons are recommended to reduce or eliminate board warpage from heat during the manufacturing process. You may change the spacing of lines in DEFAULTS of the CAM menu as well as the copper line width in the COPPER command.

5.4.1 Creating a Copper Area

The Copper command is used to draw copper areas that will be solid. When Copper is selected, the menu window is as shown in Figure 5.8.

1 New Poly	**2**	**3**	**4 Copy**	**5 Modify**
6 Move	**7 Delete**	**8**	**9**	**10 Exit**

Figure 5.8 - The Initial Copper Menu

A copper area is drawn by using the NEW POLY option on the layer specified by the Default Level and with a line width specified by the Default Line Width. This line width defines the "Spot" that will be used to draw the copper, and affects the actual plotted results of the copper.

Position the cursor at the point where the copper is to be and select New Poly. When selected a line representing the copper originates at the cursor location. As the cursor is moved, the line of the copper polygon will follow. The options used to create a new polygon of copper appear as in Figure 5.9. Use the options of Figure 5.9 discussed below.

1 Add Crn	**2 Del Crn**	**3 Angle**	**4 Level**	**5 Window**
6 Expand	**7 Contract**	**8 End**	**9 Complete**	**10 Exit**

Figure 5.9- Create New Polygon of Copper Menu

Add Crn. The Add Corner command is used to add a corner to the line. A corner is place at the cursor position when the option is selected. Select ADD CRN. The corner is constrained to be orthogonal or to the 45 degree diagonal.

Del Crn. The Delete Corner command is used to delete a corner, or back up. When selected, it will delete the last corner inserted in the copper line.

Angle. The Angle option is a toggle to allow/ disallow the current line to be drawn at any angle.

Level. The level is used to change the current level for drawing a copper polygon. "Poured Copper" should only be placed on electrical levels, however, "Copper" may be placed on levels 1 through 30.

When selected, the message:

CHANGE LEVEL [X] TO>

requests that the New Level be entered. Type the new value followed by <CR>.

Window, Expand and Contract can be used to manipulate the display window.

End. This command is available only when creating Fixed Copper. It is used to end the copper line at the current position of the cursor. This action will not create a closed polygon. This feature of Creating Copper to draw a thick copper line over a Track and then using End, is a very

useful method for thickening up tracks, placing a Ground guard around an area or clearance around the edge of a board's imbedded plane.

Complete. Used to close a copper polygon. The CRT representation of copper will be filled. During post processing of artwork, the area within the polygon will be solid copper or cross-hatched copper, depending upon the copper track width and default grid used.

5.4.2 Copying a Copper Area

The COPY COPPER option is used to copy and manipulate a copper polygon.

To use, place the cursor over the desired line of copper (or polygon edge if the copper is an area) and select COPY. The copper item will be copied and the menu will be changed as shown in Figure 5.10 as follows:

1 Complete	**2 Rot 90**	**3 Set Ang**	**4 Spin**	**5 Window**
6 Expand	**7 Contract**	**8**	**9**	**10 Exit**

Figure 5.10 - The Copper Move Menu

Use the above options to move or rotate the copper. Window, Expand and Contract can be used. COMPLETE (F1) finishes the Copy activity. EXIT (F10) cancels the command with saving the copied copper area.

5.4.3 Modifying a Copper Area

MODIFY changes the shape, line width and level of the copper polygon. When Modify Copper is selected, the Figure 5.11 is displayed in the Menu window, as follows:

1 Move Crn	**2 Move Seg**	**3 Cut Seg**	**4 Add Crn**	**5 Net Name**
6 Del Crn	**7 Line Width**	**8 Level**	**9**	**10 Exit**

Figure 5.11 - The Modify Copper Menu

The functions of Modify Copper are similar to Modify Board, except NET NAME (F5). NET NAME allows a Net Name such as GND to

be assigned to a piece of copper. When so assigned, it is a part of the Net GND. Copper that is part of a net will not cause DRC violations. Poured Copper that is part of a net will not create isolations around tracks that are part of the net, and will create thermal relief pads around pins that are in the net rather than clearances.

To use, place the cursor over the edge of the copper area, Select F5 Net Name, and respond to the prompt

Net Name >

with the Netname followed by <CR>.

5.4.4 Moving a Copper Area

The MOVE command is used to move the area of copper around on the board and rotate it. To use MOVE, place the cursor over the copper to be moved and select MOVE. The current Create Copper menu will be replaced with the menu shown in Figure 5.14.

Use the above options to move or rotate the copper. Window, Expand and Contract can be used. COMPLETE (F1) finishes the Copy activity. EXIT (F10) cancels the command.

5.4.5 Deleting a Copper Area

The Delete command is used to delete a piece of copper. To use Delete, place the cursor over the copper and confirm your decision in response to the message:

CONFIRM DELETION Y/(N)?

5.5 Flooding a Poured Copper Area

The above section discussed how to create, modify, and move a Poured Copper Area. When Poured Copper areas are created, they are drawn with there outline only. Tracks that are not part of the copper area will appear to cause violations, pads will not be isolated, and cutouts cannot be visualized. It is generally valuable to be able visualize the way the

1 New Poly	**2 New Circle**	**3 Add**	**4 Copy**	**5 Modify**
6 Move	**7 Delete**	**8 Combine**	**9 Save**	**10 Exit**

Figure 5.12 - CREATE DRAW EDITOR OPTIONS

The menus associated with the CREATE 2D command are similar to the menu commands used to Create Copper, except for COMBINE (5.7.8) and SAVE (5.7.9).

While creating 2-D lines, the Cursor Location window appears. The upper line displays the cursor position relative to the system origin, and the lower line displays the location of the cursor relative to the start of a line or the last corner in the line. The second line is of great assistance in accurately drawing specific length of lines.

5.7.1 Creating a 2-D Line

To start a new polygon, be in the Create 2-D Line option, place the cursor where you wish the drawing line to start and select NEW POLY. The Menu and operation is identical to Create Copper, with the exception that a Drawing Line can be put on any of 30 levels or layers, as well as level zero, not just on the Electrical layers.

A line is drawn starting from the cursor position and is displayed at the CRT. Use the options of Add Corner, Delete Corner, Angle to create the drawing line. If it is to be a closed polygon, select Complete (F9) to finish the last segment at the starting point. If the drawing line is not a polygon use End (F8) to terminate the line at the current location of the cursor. The 2-D Lines are displayed at their true width if the Real Width of DISPLAY is set equal to or less than the 2-D line width.

5.7.2 Creating a Circular 2-D Item

To create a 2-D circle, select NEW CIRCLE. Place the cursor on the circumference of the circle to be created (the start point) and select NEW CIRCLE. Move the cursor away from the start point. The cursor movement distance and direction determines the circle diameter and its direction from the start point. The circle is constrained to one of the eight orthogonal or diagonal directions from the start point.

ANGLE (F2) permits the direction of the circle diameter to be at any angle from the start point.

LEVEL (F4) and LINE WID (F3) permit changing of the layer the circle is on and its Line Width. When the circle is at the correct diameter and at the correct location, enter the circle into the database by selecting COMPLETE (F1).

5.7.3 Adding 2-D items from the Library

The purpose of the ADD option is to allow adding to the existing design a 2-D item that has been previously created and saved using SAVE. The item is stored in the Lines library of the Parts Library Database.

To implement, select ADD, and in response to the message:

ADD ITEM WITH NAME>

Type in the name of the saved draw line item <CR> and a copy of the draw line item will be transferred from disk into memory. If you do not know the name of the item in the library, you may use the browse feature by typing:

*** <CR>**

Scroll through the pop-up window to identify the item wanted.

ADD is a convenient way of recalling and using previously created, drawing items such as drill charts for drill drawing, dimension line arrows, etc.

5.7.4 Copying 2-D Items

The Copy command enables existing items to be copied and rotated.

5.7.5 Modifying 2-D Items

Modify Draw Line is used primarily to change the shape of the 2-D Draw Line and to change the Width and Level associated with the item.

The options contained therein are identical in function to Modify Copper, Figure 5.15, with the exception Net Name.

2-D drawing items can be placed on any level of the 30 levels, including all 30 levels. (There is no restriction to Electrical levels as in the instance of the Copper option.)

MOVE ARC (F5) permits the changing of a 2-D Line segment into an Arc and the changing of the arc radii. To use Move Arc, place the cursor over the Segment/Arc, select MOVE ARC and drag the cursor for the distance the segment/arc is to be moved. Finish by selecting COMPLETE (F1).

5.7.6 Moving 2-D Items

The MOVE Draw Line option is used to relocate or rotate the draw line item on the level on which it was placed. Move can also be used to change the draw line width as well as to transfer the draw line item to a different or all layers.

To use Move, place the cursor over the item to be moved and select MOVE. The item will be attached to the cursor and moves with it. The operation of the menu command will be identical to the Move Copper command, except that the drawing item can be placed on any or all levels rather than Electrical levels only.

5.7.7 Deleting 2-D Items

The Delete option allows the deletion of a drawing item from the database currently in memory. Note that if the item has been brought in from the library as a previously saved draw item, delete will only remove it from the current design, not from the Parts Library.

To use Delete, place the cursor over the item to be deleted and select DELETE. The item disappears and the message center provides the message:

CONFIRM DELETION Y/(N)?

A "Y" and Carriage Return will delete from the database; an "N" will cancel the Delete command.

5.7.8 Combining 2-D Lines and Text into Complex Drawing Items

Combine is used to group together into a single drawing item a number of existing drawing lines and free text (not component names or part names) that are currently in the design. After being combined, they can be manipulated (moved, rotated, copied, level changed, etc.) as if they were a single drawing item.

To add an item to the group, select ADD ITEM. Move the cursor over the first item to be added to the group, then select it. The color will change to white, indicating it has been selected for combining. Move the cursor to the second item, use Select and the cursor picks the second item and its color will also change. Continue the operation until all drawing items and text that you wish to be grouped or combined into a single group, are so grouped. After Leaving COMBINE, the normal manipulation of a single drawing item, such as Move, Rotate, etc., is applicable to the combined group of drawing items.

Groups are associated unless explicitly deleted from the group. DEL ITEM (F2) is used to delete an item from a group of items after the items have been previously combined into a group. To use, select DEL ITEM, place the cursor over the item to be deleted from the group and use SELECT. This item will be deleted from the group. The remaining items within the group will continue to remain combined as a group unless also deleted.

5.7.9 Saving a Drawing Item in the Parts Library

Save Draw Lines is used to transfer to disk for storage, a drawing item, or combined group of draw lines and text, that you expect to use repeatedly. Examples might be an inner board restriction outline of an identical family of boards, a drill hole chart, dimension arrows or any drawing feature that has a long life expectancy. To use, place cursor over the 2-D Line or combined group of lines and text and select SAVE, and in response to the message:

SAVE ITEM WITH NAME>

type in the name used for Saving the draw item. Any number of alphanumeric characters up to a maximum of 8 is valid. The Draw Line items are stored in the Subdirectory PLIB. In the user area, the text defining the Draw Line may be viewed in the Parts Library when the draw line data entry is recalled (See Chapter 12).

5.8 Creating Part Decals

PADS-2000 uses Parts that are contained in the Parts Library database. The purpose of the PART DECAL command is to graphically create or edit the Decal associated with a Part stored in the System Parts Library (LIB).

The creation of a new Part is a two-step process:

1. The creation of the Part Decal defining the physical part outline and pads. Part Decals are created using the CREATE menu, as described in this chapter and section.

2. The creation of the Electrical data describing the Part Type including pin types, pin signals, and part technology. The Electrical data is referred to as Part Type data and is alphanumeric text in content. It is created with the PLIB Library Manager as described in Chapter 12 or in PADS-Logic.

The Part Decal main menu is shown in Figure 5.13 .

1 Create	**2**	**3**	**4 Modify**	**5**
6	**7**	**8**	**9**	**10 Exit**

Figure 5.13 - CREATE PART DECAL

This menu provides the capability of either Creating a new Part Decal or Modifying an existing Part Decal contained in the Library.

When either option is selected, your board design will be saved to disk and you will be asked for the name of the Part Decal to create or modify.

PART DECAL NAME (CR =TEST)>

Enter the name you want associated with the Part Decal followed by <CR>. Use a maximum of 16 characters.

If you choose Modify, and the part decal does not exist in the library, a message will appear at the prompt line:

101 Can't find part decal item

and the board design reappears in the database and you will return to the menu illustrated in Figure 5.13. If you do not know the exact name, you can use the wild card facility and select the part by browsing.

The information below describes creating a part. The commands are similar for modifying a part.

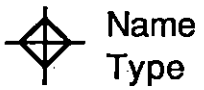
5.8.1 Creating a New Part

After providing the Part Decal name, and the menu is changed to the menu shown in Figure 5.14.

1 Outline	**2 Move Name**	**3 Terminals**	**4**	**5 Setup**
6	**7**	**8**	**9 Save**	**10 Exit**

Figure 5.14 - The Create Part Decal Menu

The display will contain a Dot Grid and the following symbol:



Name will be the default location of the Reference Name in the layout, e.g. U1. Type will be the default location of the Part Type in the layout, e.g., 7400

is the Part Decal Origin.

Creating the Part Decal consists of constructing the Part Decal Outline, entering the pin/pads, or copper at the appropriate location, moving the part origin coincident with Pin 1, and if necessary, relocating the name/type relative to the part outline, setting the pad stack size, shape and drill hole, and finally, upon completion, saving the new Part Decal in the Parts Library.

5.8.2 Creating the Decal Outline

The Part Decal Outline is constructed to indicate the actual shape of the Part. To start the Part Outline, select OUTLINE. The menu will change to as shown in Fig. 5.15.

1 Text	**2**	**3 Copper**	**4 2-D Lines**	**5**
6	**7 Cut Out**	**8**	**9**	**10 Exit**

Figure 5.15- The Part Decal Menu

The 2-D LINES (F4) command is used to create the Part Decal outline. The Part Decal can contain multiple outlines on different layers. We recommend that you place the outline on Layer 0. This way, the system will recognize the outline and place it on the appropriate silk-screen layer, top or bottom, when the component is place on the top or bottom of the board.

When creating an outline on a layer, insure that the LEVEL indication in the upper left corner of the display has been set to the level on which the outline is to be created.

When the part has a component outline on more than one level, the proper level must be made visible using the DISPLAY command of SETUP, and making 2-D Lines visible on the appropriate level.

To create the component outline, select 2-D LINES (F4). The menu options are similar to those of the 2-D Menu in Figure 5.12. And the operation is the same. Multiple outlines on different layers are allowed.

5.8.3 Copper in Decals

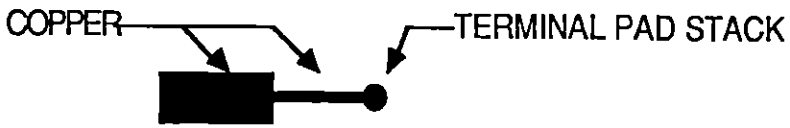
The COPPER (F3) command is used to create areas of copper on a component. The Copper that is added is fixed, not Poured Copper. It will not allow routes or other tracks to go through the area. There are three purposes for copper:

- to define complex pad patterns.
- to create heat sinks or routing obstacles under components.
- to create SMD paste masks.

The first and third purposes are very clear, but using copper to define pads needs further explanation. In the example below, copper has been used to construct the SMD pad areas, then to connect the pad areas to the Terminal PAD STACK, which penetrates to the internal layers. (The terminal was created with the ADD TERMINAL command). When copper is used to extend the terminal pad configuration, it should also be given the same name (number) of the terminal with which it is associated, using (F5) PIN NUMBER of the COPPER MODIFY commands. The purpose of naming the copper the same as the terminal pad is for use in the DRC Checking menu, where the copper area and its terminal pin are to be electrically equivalent.

NOTE: After copper areas have been named, the selection of a terminal will result in the areas of copper using the same terminal name, being displayed using the copper areas outline and not filled.

When the copper area is given the same name as the PIN/PAD, they become electrically equivalent. This allows design rule checking of copper, performed in the CHECK command, on the copper in the same fashion as if it were a pad and part of a "connection net."



The operation of the commands in the Component Copper menu are similar to those found in the Copper command defined in Section 5.4, and will not be explained here.

NOTE 1: When copper is used in a decal pad, it is ALWAYS plotted as a SOLID area, using the line width selected at the time of creating the decal pad copper area.

NOTE 2: When creating a component decal copper pad that is a square or rectangle (each side is either Vertical, or Horizontal) the size of the copper area will be EXACTLY the same size as the copper area constructed with the mouse cursor. The system considers the size of the aperture used to create the copper and keeps the area within the sides of the rectangle or square.

NOTE 3: When constructing narrow areas of copper, insure that the combination of Line Width and Copper Area Width are possible combinations. For example, a .020" Line Width (Aperture) could not be used to construct a .015" wide copper area. This will cause the message "Failed to Shrink Copper" to appear.

NOTE 4: When copper is used to create portion of a component terminal pad, it should be given a name that is the same as the terminal pad number, i.e.; copper associated with Pad Number 2 should be given the name of 2. To do this, proceed as follows:

- First place the terminal.
- Construct a copper area that defines the pad. In our example, this copper area was created as two areas, a polygon and a copper line.
- Attach the copper area(s) to the correct terminals.

5.8.4 Adding Text to the Decal

The TEXT command (F4) is used to add free text to the Decal. Examples of Free Text are a + (Plus), at the cathode end of a diode, or 1 to indicate pin 1, or polarity of a component. To use, select TEXT (F1), of the OUTLINE command, then CREATE (F1), then enter the desired Text, and place it at the correct location.

Decal text is treated as "free text" when the decal is brought on to the board and is displayed at the CRT, and plotted as normal free text.

5.8.5 Creating a Copper Cutout

It is possible to define a copper cutout in a part with the command CUT OUT (F7) . When selected, you can draw a cutout area in the same fashion as cutouts are created in the CREATE menu. When the part is used in a layout, the cut out is transferred into the layout and will automatically cause a cut out to appear in a Poured Copper area.

5.8.6 Moving the Default Part Name Location

The command of MOV NAME (F2) is used to move and rotate the Part Decal Text which defines the default position of the Reference Designator and Part Type.

When MOVE NAME is selected, the Name and Type are attached to the cursor and the standard PADS-2000 Text Move menu is presented.

Move the name relative to the Part Decal outline, rotate if necessary and place using COMPLETE. Incidentally, the word "name" is the space reserved for the Reference Designation of the part when used on the board, i.e., U1, R6, etc. The word "type" is a space reserved for the type of component, i.e., 7400, 7404, etc.

5.8.7 Placing Terminals Pads

The TERMINALS command (F3) is used to create and locate the pads or pins of a Part Decal and assign the correct pin number, size, and shape to each pad.

When the TERMINALS command is selected, the menu changes to that of Figure 5.16.

1 Add Term	**2 Move Term**	**3 Move Name**	**4**	**5**
6 Pad Stacks	**7 Del Term**	**8**	**9**	**10 Exit**

Figure 5.16 - The Create Terminal Menu

To add a Terminal to the Part Outline, select ADD TERMINAL. A filled white circle representing the Terminal is attached to the cursor. The

Terminal may be moved, and the final location of the terminal is selected by moving the cursor to the desired location of the terminal and selecting COMPLETE (F1). The white circle will be changed to a red filled circle where the terminal is to be located. The Terminal pad number will become visible next to the terminal pad location. The terminal size and shape are indicated.

Terminals are numbered as they are added, starting from 1. The numbers are incremented automatically; therefore, terminals should be added in the sequence specified by the part vendor's specification drawing, i.e., Pad #1 is located first, followed by pad #2, followed by pad #3, etc.

To move a Terminal after it has been placed, place the cursor over the terminal pad and use MOVE TERM (F2). The terminal will be attached to the cursor and can be relocated and placed with COMPLETE (F1).

To move the Terminal Name relative to its pad, place the cursor over the Terminal Number and select MOVE NAME (F3). The Terminal Number will be attached to the cursor and can be relocated from its original Default position at the upper right quadrant of the pad. Move the number to the desired location and select COMPLETE (F1).

To delete a Terminal, place the cursor over the terminal and select DEL TERM. After confirming the deletion, the selected Terminal will be deleted. All terminals with numbers higher than the number of the deleted terminal will have their terminal numbers automatically deincremented.

The option of PAD STACKS is used to generate the pad stack criteria (size, shape, drill, etc.,) for each pad on a Part Decal. Use PAD STACKS only after all the terminal pads for a Part Decal have been added.

The operation for defining Padstacks is given in Section 4.2, explaining the PADS padstack definition command.

5.8.8 The Setup Menu

This function is similar to the SetUp Menu in the design program. Commands control the color display and other parameters.

5.8.8.1 Controlling the Display

The colors assigned to various items in the Part Decal are set with the Display menu. This is similar to the Display Window of the Setup command (see Section 4.1 for details), with the following changes:

Certain items, e.g. Tracks, Vias, Connections, Errors, Top, and Bottom are not available as they do not have meaning while creating a Part Decal.

Part Names and Part Types are assignable only on the top layer.

5.8.8.2 Changing the Origin of the Decal

The Origin associated with the Part Decal is not the same, nor does it have any relationship to the System Origin of the board database discussed previously in Chapter 4, SETUP. Instead, Origin, within the Create Part Decal command applies ONLY to the construction of the Part Decal. The Part Decal Origin appears to the left of the name/type text thus:

When the decal is used in a design, the part is selected by cursor on the origin, and moved, placed and rotated about this origin. This origin is also used when a report giving the location of the part in the layout is printed. It therefore is wise to place the Part Decal Origin in a meaningful place. Typically this will be pin 1 of the component, but you may choose to place it elsewhere.

To relocate the origin of the Part while making a Decal, move the cursor to the location where you want to position the origin and select ORIGIN (F2). The Part Origin will be relocated to the cursor location.

5.8.8.3 Changing the Display Dot Grid

The Dot Grid is a useful visual tool in scaling and drawing of Part Decal Outlines and locating the terminal pads. Should you wish to change the Dot Grid spacing, select DOT GRID, and respond to the message:

DOT GRID SPACING [100]>

with the new spacing followed by <CR>

5.8.8.4 Max Layer (Multilayer Part Decal)

MAX LAYER is used during the creation of a Part Decal, when it is known that a Part Decal will be used ONLY on a multilayer board. Normally the Part Decals are created with only 2 levels, which can be used on both 2 layer as well as multilayer boards. The Pad Stack data is then set for the multilayer usage during the design phase using SETUP and PADS.

If you know that a Part Decal will be used only for a multilayer board, then during the creation of the Part Decal, before creating the terminals, select MAX LAYERS and respond to the message:

MAXIMUM ROUTING LAYER [2]>

Respond with the maximum layer number of 4. The default pad setting for a 4 layer Part Decal will then be stored with the Part Decal.

NOTE: If you have a 2 layer version and a multilayer version for a Part Decal, you will need to distinguish between the two versions, in the name assigned to the Part Decal. In addition the Part Type will need to distinguish which Part Decal to use. You can assign both to the same part type as alternates

5.8.8.5 Setting the Units

PADS-2000 allows you to create your parts in either metric or inch coordinates. UNITS is used to establish the dimensional units of the system grid during the Decal creation. Either Imperial or Metric units can be used.

To establish the dimensional units, select UNITS (F5) and respond to the prompt:

Units are Metric, Mils or Inches [Mils] (ME/MI/I)>

ME <CR> results in a Metric unit system grid.

MI results in a grid with .001" units

I results in a grid of inches.

Decals are stored in PLIB in the fundamental system grid, which is less than 10^{-8} meters, regardless of the units of construction. This insures that no dimensional accuracy is lost whether it is used in a metric layout or inch layout.. When a Decal is called from PLIB to be modified, or used in the Board design, it is always brought back into the board in the coordinate system of the design, regardless of the units used in construction.

A Decal that has been created using either Inch or Metric units can be used on a board using either Inch or Metric units.

5.8.9 Saving the Decal

The command of SAVE, is used to transfer the Part Decal that has been created to the hard disk for permanent storage in the Part Decal Library.

Select SAVE and the message:

PART DECAL NAME (CR=XXX)>

appears, where XXX is the original Part Decal name given at the time of starting the Part Decal. If you wish to save it under the name used when starting the Part Decal, strike Carriage Return and the part will be saved. The Display "Working..." appears for a few seconds while the part is being written to hard disk. The Part Decal is stored in the Parts Library located in the directory \PADS\LIB.

The PADS Part Library has two areas: (1) The User area and (2) the Standard area. The Part Decals created and "Saved" by the user using SAVE are stored in the User Part Decal Library, NOT the Standard Part Decal Library The Standard Part Decal Library contains those Part Decals supplied with the original software.

5.8.10 Modify an Existing Part Decal

It is possible to change an existing Part Decal graphically. Modify Part Decal is used to modify the outline, relocate name/part type block location, change pad and change pad and relocate pad number location on decals previously created. To use Modify, be in the Part Decal command and select MODIFY. The message:

PART DECAL NAME (CR=TEST)>

asks for the name of the Part Decal to be modified. After entering the Part Decal name, the menu will change to the menu of Figure 5.14, and provides all of the functions that are used while creating new Decals.

When saving the Part Decal after Modify, if you use the same name to save the part as was used when originally created, you will be asked the question:

PART DECAL ITEM ALREADY EXISTS <XXX>
OVERWRITE LIBRARY ITEM Y/(N)?

If you wish to retain both the original Part Decal, as well as the modified Part Decal in the library.

5.9 Part Types

(Refer to Section 12.6)

CHAPTER 6 - PLACING PARTS

The objective of this chapter is to familiarize you with the operation of the commands in PADS-2000 Place menu for component placement.

The Place menu is used to place Parts on the board. The Editor has ten functions:

- To move a Part, using the cursor, to any location on the board at a resolution as determined by the Default Grid setting.
- To move a Part to a point on a user defined Placement Matrix.
- To rotate a Part.
- To move a Part from the front side to the opposite or back side (such as in the use of SMD's).
- To move the name of a Part, relative to the Part outline.
- To "glue" a Part down on the board so that it cannot be moved.
- To "unglue" a Part that has been "glued" so it can be moved on the board.
- To create, save, move groups of parts.
- To use the autoplace functions to automatically place and rename components.
- To swap gates, parts, and pins with full Backward ECO capability.

When the PLACE command is selected, the main Menu is replaced with the commands of the Place Menu shown in Figure 6.1:

1 Move	**2 Mat Move**	**3 Rotate**	**4 Opposite**	**5 Glue**
6 Unglue	**7 Move Name**	**8 Auto**	**9 Group**	**10 Exit**

Figure 6.1 The Placement Commands

6.1 Move vs.Matrix Move

MOVE is used to move a Part to any location on the board, as defined by the resolution of the System Grid. During Move, the Part can be placed with a resolution equal to the current system grid. For example, if the Grid and Units were set at .100 and Inch, the cursor would move in units of .1". The part would snap to the nearest .100" grid point.

MATRIX MOVE, as distinguished from MOVE is used to move a Part to the near vicinity of a point on the PLACEMENT matrix (as created in AUTO PLACE) , and upon COMPLETE, the Part will "snap" to the nearest Placement Matrix point.

The difference between MOVE and MATRIX MOVE is: MOVE permits the movement of a Part in accordance with the System grid. MATRIX MOVE places the Part only on the matrix points of a pre-defined Placement matrix.

The options of MOVE and MATRIX MOVE are the same, and are described below.

Parts can be selected in three different ways:

1. By Reference Name

Enter the Part Reference Designation. If the Part is unglued, the Part will attach itself to the cursor at its origin location. If glued down, the message:

COMPONENT XX IS GLUED DOWN

is given. Otherwise, the component is attached to the cursor.

2. By Cursor Select

Place the cursor over the Part outline or one of the Pins of the Part to be moved and choose SELECT. The cursor automatically picks the Part Origin. If glued down, the message:

COMPONENT XX IS GLUED DOWN

is given. Otherwise, the component is attached to the cursor.

3. Global Part Move

It is possible to move multiple parts with a single command with the "*" wild card facility. Enter "n* <CR>" where n is the first character or characters in the Reference Designator name to move all parts that begin with the characters. For example, U* would select U1, U2, U3, etc.; CR* will select CR1, CR2, CR3, etc.; R1* will select R1, R10, R11, R100, etc.

The first unglued Part of the indicated family will be selected for MOVE. After MOVE of the selected part is finished by COMPLETE, the next sequentially numbered Part of the family will be selected, without action by the designer. Auto Increment will continue until all Parts of the family are selected, or until the action is terminated with ABORT.

4. Part Type

It is possible to move all parts of a given type by typiung the Part Type. For example, typing "7400" will cause each of the 7400 parts to be selected.

5. Technology

It is possible to select all parts of a given technology by typing the technology name. For example, typing "TTL" will cause all parts with the technology type TTL to be affected by the command.

If the lower half of Information Window is visible, it will display the following information:

Reference Designator
Part Type
Technology
Part Decal.

If the Part is picked by selecting one of its pins with the cursor, the pin number (but NOT the Pin Name, if a Pin Name is used) is also displayed.

When a Part is picked with the cursor, the SELECT menu is replaced with the PLACE, MOVE, window shown as follows:

1 Complete	**2 Rot 90**	**3 Opposite**	**4 Length Min**	**5 Alt**
6 Set Angle	**7 Spin**	**8 Matrix**	**9 Abort**	**10 Exit**

Figure 6.2 - Move Component Menu

The Part is moved with the mouse. If the Cursor Position window is visible, the X and Y value is the distance from the System Origin, and the DX and DY values are the distance relative to the last position of the Part.

The Part may be moved using the S Modeless command where XXX and YYY are the new location coordinates. The Part will be moved so that the Part Origin falls on XXX YYY. After the Part is moved, use COMPLETE (F1) to set the part. See section 2.4 for details.

6.1.1 Rotating a component while in the Move menu.

Components can be rotated while being moved in three ways:

1 To rotate the component in 90 degree increments about the origin, select ROT-90 (F2). Each time this is selected, the component will rotate 90 degrees counter-clockwise.

2. To rotate the components to a fixed angle, SET ANGLE (F6). You will be prompted with

Enter component rotation angle (0-359)>

The angle may be specified in 1/10 degree increments, but must be a positive value in the range of 0 to 359.9 degrees. Respond by typing the angle rotation in degrees in the format XX.X.

3. The angle may be set by moving the cursor with the SPIN command. When selected, the component is anchored at its origin, and will rotate about this origin on the cursor. The angle of orientation is displayed in the Coordinate Display Window.

6.1.2 Placing a Component on the Opposite Layer

It is possible to position components on the back of the board, and to swap them to different sides with the OPPOSITE command. Once on the opposite side of the board, the Part can be moved, rotated, etc. This option is useful for SMD designs, where Parts are located both on the component layer or opposite/solder side.

When in OPPOSITE and a component is selected, the component is moved to the maximum or highest routing layer as established by the Max Layer parameter. All of the information on the top layer is converted from the Component (top) side to the Solder (bottom) side, i.e., and the part is mirrored to reflect its view on the back of the board.

To distinguish which layer the swapped Part is on, take care to have the outline of the Part set to different colors on the component and solder layers.

6.1.3 Displaying the Minimum Connection Length

The LENGTH MIN command reorders the nets associated with the Part being moved to minimize the length of connections associated with the Part. This option allows the user to move a Part to different locations, run LENGTH MIN on the connections associated with the Part being moved, and by visual inspection, determine whether one particular location is better than another, based on connections.

NOTE 1: Connections whose Reconnect parameter is set to Off in the Net Attribute menu will not be affected by the Length Minimization function. If you are trying to run Length Minimization and nothing happens, check that the appropriate connections are set to ON in the Netlist Display.

NOTE 2: Connection nets with a very large set of pins, e.g. Power and Ground, take a longer time to calculate the Length Minimization length than normal connections. Unless you specifically want a Length Minimization of Power and Ground, the Reconnect parameter for these connection should be set OFF in the Net Control (Paragraph 4.3.4) and the length minimization will run very quickly.

6.1.4 Working with Alternate Parts

A Part Type may have as many as four different decals associated with it. For example, a resistor may have a decal for horizontal placement or it may be turned up on end. Alternately, an IC may be created in direct soldering to the board or for insertion into a DIP socket.

To provide this capability, up to four different decals using four different decal names may be created and saved. In the Part Type library, the part type must call out the Primary or first Decal (R 1/4WA) and the alternate decal names separated by a colon, in the first line of the Part Type data thus:

R1/4W R 1/4 WA:R1/4WB:R1/4WC:R1/4WD

Next, when placing the resistor on the board, the use of the ALT command will step the decal used in the database thru the primary decal then the three alternate decals, in the order of the Part Type Data Line above.

The ALT command permits the selection of up to four different Part Decals associated with a Part Type, if the Part Type has been created with multiple Decals. While a part is selected for MOVE, select Alternate Decal and the first alternate decal in the part type description will be substituted for the primary Decal, and displayed at the CRT. Repeated selection of ALT selects the other alternate Decals. If no alternates are available, or all alternates have been chosen, the software cycles back to the initial part.

NOTE: When creating alternate decals for a part, they ALL MUST have the same number of terminal PADS/PINS.

6.1.5 The Matrix Command in Move

The MATRIX (F8) command in MOVE or MATRIX MOVE is a toggle switch. If in MOVE and moving a Part, and MATRIX is selected, the status changes to MATRIX MOVE, and the part is placed on an existing matrix point. If in MATRIX MOVE and MATRIX is selected, the status is changed to MOVE. This toggle allows this type of move, without having to exit one command and re-selecting the next.

6.1.6 Abort or Exit (While in Move)

These commands cancel the movement of the selected Part. It returns the Part to its original location, and reverts the menu back to the Select/Exit menu of Place, Move. If Length Minimization has been run while the part is being moved, ABORT (F9) will undo the length minimization, thus leaving the design as it existed prior to the time the part was moved.

6.1.7 Completing the Placement

When the Part is at the proper location and orientation, use COMPLETE (F1)to final place the Part.

6.2 Rotating Components

The Rotate function allows the rotation of previously placed Parts. It operates as explained above, except that the mode for rotation, 90 degree, Set Angle, or Spin, is selected first, then the Part or Parts to be rotated are selected.

In addition to rotating a single Part, the Rotate option can also be used to globally rotate groups of Parts. These groups can be defined by their Reference Designation, Part Type or Technology.

To Rotate Parts globally by name, type in the letter of the Part Type of the Reference Designation group to be rotated, followed by *. For example, U* <CR> will rotate all IC's, assuming IC's are designated with the Reference Designation "U." U1* will rotate U1, U11 through U19, etc. R* will rotate Resistors, etc.

To rotate by Part Type, in response to the SELECT of the Rotate command, type in the Part type, i.e., 7400 <CR> and all 7400 type IC's on the board will be rotated.

To rotate by Technology, in response to SELECT of the Rotate command, enter TTL, CMOS, etc. and the group of Parts utilizing that technology will be rotated.

1 Complete	**2 Rot 90**	**3 Set Angle**	**4 Spin**	**5 Window**
6 Expand	**7 Contract**	**8**	**9**	**10 Exit**

Figure 6.3 - The Move Name Menu

Move the Name to the location desired and place with Complete (F1). The Part Name may be rotated independently of the Part rotation.

6.6 Autoplacement

PADS-2000 has an extensive set of Autoplacement tools that can be used to help define the component placement quickly. These are contained in the AUTO command. When selected, the menu changes to Figure 6.4.

1 Matrix	**2 Net Attr**	**3 Conn Bias**	**4 Init Plc**	**5 Optimize**
6 Discrete	**7 Len Min**	**8 Show Plc**	**9 Rename**	**10 Exit**

Figure 6.4 - The Autoplace Menu

The autoplacement commands need to determine which parts have been placed and have influence on the placement of other parts, and which parts are unplaced, and have no influence on placement. Since board outlines can be quite complex in shape, it is difficult to determine which parts are actually on the board and placed, and which parts are off the board and unplaced. Hence, these commands use a simple approximation to the board outline which is the smallest rectangle that encloses the entire board outline. We call this the Bounding Rectangle. Any parts inside this Rectangle are considered placed, and any parts outside are considered unplaced. It is possible with an irregularly-shaped board, that some parts that are actually off the board, but fall within the "bounding rectangle" will be considered placed by the AutoPlace commands.

6.7 Creating Placement Matrices

Placement Matrices are used to manually or automatically place the origin of Parts on a matrix point, to create a regular placement of IC and other parts. It is an aid for both manual and automatic placement.

PADS-2000 supports multiple matrices, of varying sizes and spacings. Matrices can have unusable points. A Matrix is created by the intersection of vertical and horizontal lines

The MATRIX command is used to create, modify, and select matrices.

When Matrix is selected, the menu changes to Figure 6.5.

1 Select	**2 Unselect**	**3 Create**	**4 Delete**	**5 Move**
6 Del Pnt	**7**	**8**	**9**	**10 Exit**

Figure 6.5 - SET MATRIX OPTION

SELECT (F1) selects, from the previously created Placement Matrices, an Active Matrix. Multiple Active Matrices is allowed, and will be used when a command making a placement of Parts on a Matrix is executed. Active matrices are visible.

UNSELECT is used to make an Active matrix inactive. The cursor is used to select the matrices to turn off. Inactive Matrices are invisible.

CREATE is used to define a new Placement Matrix. A matrix is created by responding to four questions presented at the CRT Message Window. These four questions are:

Number of rows?
Space between rows?
Number of columns?
Space between columns?

Rows is the number of Horizontal lines on the Y axis. Columns is the number of Vertical lines on the X axis. The response to the above creates a Placement matrix of a user specified number of rows and columns with the user specified spacing. The intersection of each row and column is a point on the matrix. The lower left hand matrix point is at the location of the Mouse cursor at the time the Placement Matrix was created. The Placement Matrix is attached to the cursor and can be moved about on the board and final placed using COMPLETE.

A Placement Matrix that exceeds the 56" x 56" database size is not accepted for use. Spacing between rows and columns must not be less than .1". A maximum of 50 rows and 50 columns is possible in a single matrix.

The DELETE (F4) command deletes selected Placement Matrix. The message:

CONFIRM MATRIX DELETION Y/(N)?

allows the designer to confirm the intention of deletion.

A Matrix may be moved without changing its size or spacing with the MOVE (F5) command. The option permits a created matrix to be moved around on the board, or placed on the opposite layer. The color of a top and bottom Matrix will be different.

It may be desirable to not allow certain points of a matrix to be used during placement. They may overlap a cut out in the board, or be part of a restricted area in the circuit. It is possible to specify certain points as not usable with the DELETE POINT (F6) command. To use, place the cursor over the point to be deactivated, and select DELETE PNT (F1). A circle will be created and centered around the deleted point, indicating the matrix point is not active.

To restore, or reactivate, a deleted point, place the cursor over the center of the deleted point and select DELETE PNT again. The formerly deleted point will be made active again and the delete point circle will be erased.

6.8 Placing Components on a Matrix

There are two commands used to place components on a matrix, Matrix Place and Initial Place.

Matrix Place is used to automatically place unglued Parts on the Placement Matrix in a "random" fashion. The Part Origin of each Part falls on matrix points of the Placement matrix. It merely serves the purpose of getting the parts upon a matrix on the board ready for you to complete. If the number of unfixed Parts exceeds the number of points on the Placement Matrix, the matrix will be filled up and the excessive

unplaced Parts will remain unplaced. Placing Parts on the matrix using MATRIX PLACE allows the designer to see all the Parts and their connections and final place the Parts by using the connection rubber band display as a guide.

The INIT PLC command is also for placement of IC's on a matrix. It works similarly to the MATRIX PLACE command, except that it is automatic, and will calculate an approximate best location for all unglued components on the matrix based on the connectivity of the circuit. It results in significantly better initial placement solution than is achieved by using the Matrix Place command.

The INIT PLC command works on all unglued parts, whether on or off the board. The command will tend to place any part that is heavily connected to a pre-placed part, as close as possible to that part. Any part that is glued down and contained within the "bounding rectangle" of the board is considered to be pre-placed, and will influence the placement of other parts connected to it. Any parts, such as discrete parts, that are not to be placed with this command should be glued down outside the "bounding rectangle," or they will influence placement where not intended.

The recommended steps to take before using INITIAL PLACE are as follows:

- All "fixed location" parts, such as connectors, have been placed and glued down. (mandatory).
- Any large irregularly-sized parts, such as microprocessors and gate arrays have been placed and glued down. (optional, but highly desirable)
- Any large memory arrays have been placed using Autoplace commands and glued down. (optional)
- A matrix has been defined for the remaining IC's. (mandatory)
- All other parts that are not to be placed on the matrix are glued down. (mandatory)

After selection, the command will display the message:

Performing "Set-up Phase"

on the message line, after which the message:

Placing Components

is displayed during which a busy message will count down the number of parts to be placed. After all the parts have been placed, the message:

Updating Database

is displayed, during which a Length Minimization is performed. Upon completion, the final connection length is displayed.

NOTE: If it is not known where the large irregularly-sized parts are to be placed; they can be placed along with the other IC's by the Initial Place command. However, this command does not consider the actual size of a part, and will allocate one part per matrix point. Hence, large parts will likely overlap parts on neighboring matrix points. Once the positions of the large parts have been determined, they can be glued down, and the matrix points which they cover can be de-selected. Initial Place can then be run again to place the other IC's around the large parts.

NOTE: If the placement area for the IC's is not a conveniently-shaped rectangular region due to the shape of the board, or due to a pre-placed memory array, a matrix should be defined to enclose the desired placement area; and the matrix points not in the desired area should be de-selected. The user should then ensure that there are sufficient matrix points remaining to accommodate the IC's to be placed.

NOTE: If the design contains many Surface Mount Devices and placement of these parts is desired on both sides of the board, then a "mirrored" matrix should be created for the bottom side of the board in addition to any matrices defined for the top side. Additionally, those parts to be located on the bottom layer should be moved to "Opposite" before AUTOPLACE. Surface Mount parts can be placed on either a top side or bottom side matrix depending on which has the best location for the part. Parts that are not Surface Mount devices are placed only on top side matrix points.

NOTE: If the circuit is predominantly digital, power and ground nets should be made invisible using the Net Attr command before running any of the Autoplace commands. Leaving them visible will cause them to affect the placement and may result in a poor placement as well as take a long time to complete.

NOTE: If there are more parts unglued than there are matrix positions available, the parts left over remain in their original locations.

6.9 Optimizing the Placement by Swapping

The command of OPTIMIZE in the Autoplace menu permits the automatic swapping of three items as follows: (1) Parts, (2) Gates and (3) Pins. These are described in detail below.

6.9.1 Swap Parts

SWAP PARTS is used to automatically swap all parts or manually swap two selected Parts with each other. When selected, two modes are available, AUTO and MANUAL.

The AUTO command is for fine tuning of IC placement after the IC's have been placed on a matrix. It swaps each part with each of its neighbors to see if the swap improves placement, by reducing the total connection length and biasing rules.

The command will swap each part in the circuit with each of its unglued neighbors. If the swapped position results in a reduction in connection length, the parts stay swapped. Otherwise, they are swapped back to their original positions, and an attempt is made to swap the part with the next neighbor. If a neighboring matrix point is empty and it is not de-selected, the command will swap the part into the empty location to determine if the new position results in a reduction in connection length.

Before running the AUTO command, the following items should have been performed:

- A matrix has been defined and Parts have been placed on the matrix with the Initial Place command, or manually with the Matrix Move command.
- All parts on the matrix to be considered by this command should be unglued, and all parts on the matrix that should remain fixed are glued down.
- The Biasing Rules are established. See section 6.11 for details

When the command is selected, the message:

Performing "Set-up Phase"

appears on the message line for a short period of time (no more than a few minutes) after which the message:

Iteration #N Pairs Swapped: YYY

will be displayed, where
N indicates which iteration is being performed and
YYY will be the actual pairs swapped during the N iteration.

The command may perform several iterations until an iteration produces no further reduction in connection length and no pairs are swapped. If, at any time the user feels that the rate of reduction in connection length between each iteration has reduced to a point where it is not worth waiting for the command to finish normally, he can terminate the command by selecting the key F10.

When terminated through F10, or when no further reduction in connection length has been obtained, the command will display the message:

Updating Database

during which a final Length Minimization is performed. Upon completion, the final connection length is displayed.

The command displays the connection length before and after the swap is performed, and then allows the designer to accept or reject the swap results.

To use MANUAL command, place the cursor over the first of the Parts to be swapped, and select MANUAL. The first Part is selected (activated) and the menu changes to a "Select/Exit" Menu The designer next selects the component to be swapped with the first selected Part. The second Part is highlighted to indicate it has been selected; the two Parts are then swapped.

The "before" and "after" X, Y and X + Y connection lengths are displayed on the Prompt Line to assist the designer in the decision to accept the swap. If acceptable, CONFIRM (F2) is chosen to set the

components in their new locations. If EXIT (F10) is selected, the two Parts to return to their original locations.

6.9.2 Swapping Gates

This command provides for automatic or manual swapping of gates in related components. It supports back-annotation by producing an ECO report and file for transfer to PADS-LOGIC, Schema, and other schematic drawing packages.

The selection of SWAP GATES (F2) results in the message:

GATE/PIN SWAP REPORT FILE NAME

Typing:

<CR>

will result in a listing of the results of gate swapping on the printer. Responding with a file name will create a file in the \PADS\FILES\ directory with the extension .ECO containing the results of the gate swapping activity. If the file already exists, a Prompt will ask:

Append File [\PADS\FILES\ZZZ.ECO] Y/N?

After responding to the message, the menu changes to offer two options, MANUAL or AUTO.

Manual Swap enables the designer to select specific gates to be swapped. Using the cursor or the keyboard, select the first pin of the gate that is to be swapped following the selection. If swappable, the connections associated with the gates selected are highlighted to indicate that the gate is to be swapped.

A second gate is then selected. To be swappable, it must be of the same Part Type as the first and not be glued down. If the second gate is swappable, its connections will be highlighted and you are requested to CONFIRM (F1) the swap. The selection of CONFIRM (F1)will result in a swap. EXIT (F10) will cancel the gate selections.

To select the gate by keyboard, the entry is

UU.G

where
UU is Component Name or Reference Designator, e.g. CR1, U10
G is the Gate Identifier, e.g. A, B

Possible error messages include:

GATE IS NOT SWAPPABLE

The part type data for the selected part has indicated the gate as not swappable.

GATES IN DIFFERENT PART TYPES

The two selected gates are in two different part types and hence, are not swappable.

PART IS GLUED

The gate cannot be swapped because the part is glued.

NOTE: When a gate is selected, it is identified in the Information Window by the Component Name and Gate.

Auto Gate Swapping is selected when the designer desires the gate swapping algorithm to perform the swap operation on the entire layout. When the option begins, a status report in the Information Window gives the status of the swapping operation. Gates of all parts that are unglued and are identified as swappable in the part type data will be swapped.

6.9.3 Swapping Pins

Swap Pins is used for swapping pins within the gate of a part or within the component. When Swap Pin is selected, the user is asked to define the name of the ECO file as discussed in the previous section.

The designer may select the pins of a gate or a part to be swapped with the MANUAL command The designer select the pins to be swapped using the cursor or keyboard. Note that after selection, but before

CONFIRM or REJECT, the connection pattern that would result from the swap operation are shown for the designer's inspection as highlighted connections.

A second pin is then selected. To be swappable, it must be of the same Swap Type class and Part Type as the first and not be fixed. If the second pin is swappable, its connections will be highlighted and you are requested to CONFIRM the swap. The selection of CONFIRM (F1)will result in a swap. EXIT (F2) will cancel the pin selections.

To select the pin by keyboard, the entry is

UU.P

where

UU is Component Name or Reference Designator, e.g. CR1, U10
P is the Pin Number, e.g. 1,2

Possible Error Messages are as follows:

WARNING!! PIN SWAP CLASS NOT DEFINED

The part type data does not define the rules for swapping the selected pin, however the designer can confirm and make the swap if desired.

NO CONNECTIONS

No connections to pin.

PINS ARE NOT IN SAME PART

The Pin Swapping rule for the Part Type does not permit pin swapping between parts.

NOTE: When manually swapping pins on an edge finger with pins on Layer 1 and pins on Layer 2, you may have difficulty using the cursor and SELECT to select Pins on Layer 2, consequently use keyboard I/O and identify the pins by pin number.

When Auto Pin Swap is selected, the system immediately enters the swap set-up phase (set-up identifies all unglued parts as well as pins that have been classified as swappable), after which a series of swapping

horizontal, and vertical directions. The histograms generated are only approximate, since they do not take into account the pad locations on the parts; but they do give an indication of the routing channels across the board that are likely to cause problems during routing.

Upon selecting either histogram command, the following prompt will be displayed:

Enter Routing Grid (5 - 250) [25]:

The user should enter the desired value, based on the system grid used, followed by <CR>. The default response is shown in the square brackets.

After the "Working" message has counted down the number of connections processed, the histograms are displayed with the horizontal channel histogram being displayed along the left edge of the board, and the vertical channel histogram being displayed along the top edge of the board. The peaks in the graphics indicate channels where there are many connection points. These should be examined to determine autorouting feasibility.

6.12.3 The Connection Density Map

The DENS MAP command is another way of displaying the density of connection. It divides the board up into cells, and displays the density of connections in each cell using different colors to show relative connection density. This Density Map readily identifies the connection "hot spots."

Upon selecting the DENS MAP command, the following prompt will be generated:

Enter Display Grid (25 - 500) [100]:

The user should enter the desired value, although the default of .100" or .25 mm is usually satisfactory. The default response is shown in square brackets. After the "Working..." message has counted down to zero, the color density map of the board is displayed. The colors on the map show relative density. The colors used, (ordered from highest density to lowest) are as follows:

Red (highest density)
Yellow (dense,but lower than highest density category)
Green (third lowest density area)
Blue (lowest non-zero density)
Black (zero density)

After the map has been displayed, the board and its contents are re-drawn on top of the map, so the map becomes the background.

NOTE 1: Connections that are turned off in the NET DISPLAY of SET-UP will not be considered.

NOTE 2: The track WIDTH of a connection is not considered. A .012" wide signal track is weighted the same as a .050" power track.

NOTE 3: The Density Display is relative - Not absolute. A color of RED does not indicate that all the tracks cannot be completed in the Red area. Instead, the RED indicates that this area, relative to non-red areas, has a greater connection density.

NOTE 4: Before running the Connection Density Display routing ensure that only the board items that are to be drawn on top of the map are visible (usually only the board and part outlines).

6.13 Component Placement by Group

The Group Placement commands permit the designation of an area containing parts, connections and tracks to be a group, and manipulated with single commands. Items can be added or deleted from the group and a Group can be saved to disk, and Pasted to another design.

6.13.1 Defining a Group

Groups are defined using a bounding rectangle. Items may then be added or deleted from the group area. To initially define the Group, place the cursor at the lower left corner of the items to be grouped and select DEF GRP (F1). As the cursor is moved, a rectangle tracks the cursor. Move the cursor to the upper right corner of the area to be defined and select COMPLETE (F1) to enclose the group area. All

items in the group will be highlighted, including tracks and connections that connect between two pins within the Group, Components, Text, Copper, Cut Outs and 2-D items, except the board outline.

NOTE: Items which are NOT displayed will also move if contained within a group. The User is advised to display ALL elements prior to using the Group function.

ADD ITEM (F8) permits the adding of an item (part, text, but not connections) to the group. Place the cursor on the item and select ADD ITEM (F8). The item will be highlighted indicating that it has been added to the group. When the group in moved the item that has been added will retain its position relative to the group, after group move has been completed.

DELETE ITEM (F9) permits the deleting of any item inside the group from the group. After the group has been created, place the cursor on an item within the group and select (F9). The item to be deleted will change color indicating it has been deleted from the group. If the group is now moved, the deleted item will not move with the group.

6.13.2 Working with Groups

Groups of components and related information may be moved, rotated, copied, deleted, cut and pasted, or swapped to the back of the circuit. For Move and Copy, a rectangle will be positioned on the cursor. When the final position is selected with COMPLETE (F1), the components in the Group are drawn.

In general, these commands are selected and executed similarly to their single component counterparts. Differences are noted below.

During a Move or Rotate Group command, tracks inside a group area will retain their position after Group Move. Tracks going outside the group will be stretched to the new group location.

When using Group Copy, only those connections and tracks that are between two pins that are both members of the Group will be copied. Component names are automatically incremented to unused names, to insure that duplicate names will not exist after the COPY is made. Connections that are copied are given new $$ signal names.

DELETE will delete the contents of the Group, after responding to the confirm prompt. The effect on nets with members outside the group is similar to that when Del Pin is used.

OPPOSITE acts to "mirror" the group about the y-axis, and will transfer the components and tracks on the top layer to the bottom, and from the bottom layer to the top.

6.13.3 Using Groups in Other Designs

PADS-2000 has the capability to save a group of components and their associated tracks to the hard disk for use in another design. This is particularly useful for using a subcircuit, e.g. power supply or address decode, on several boards in a system. The CUT command will save the group to disk, as a separate file. PASTE will load a group from disk into the current design.

CUT is used to save the contents of the Group to hard disk as a file. This file can be recalled and used in the same board on an entirely different board. When (F6) CUT is selected, the following prompt is given:

ENTER FILE NAME>

Respond with the name that you wish to save the group under, followed by <CR>. The group will be saved with the name and extension .CLP in the PADS\FILES directory.

PASTE permits the "pasting" of a group file that has been previously saved into the current design. To use, select PASTE (F2), and respond to the prompt:

ENTER FILE NAME>

with the .CLP file name used to CUT and SAVE the group to Disk. The CUT is recalled from the PADS\FILES and placed at the cursor location. The names of the components in the Pasted group are automatically incremented so that there are no duplicate part names. Completed tracks within the cut are reproduced. Tracks that originate or terminate outside the cut are dropped. Reproduced tracks and

connections are automatically given a different signal name, so as to not duplicate existing signal names.

6.14 Auto Rename of Components

This command is used to renumber Reference Designators by row, or by column, or both. This command is normally used after placement has been finalized. This command assumes that each reference designator is made up of an alphabetic "prefix", such as "U", or "CR" and a numeric "suffix." Only the numeric suffix is altered by this command.

Before running this command, a matrix should be defined and selected for the sole purpose of defining the rows and columns for renaming. (This is not necessarily the same matrix used for placement; although on some boards, it may be. In general, the Rename Matrix should have more matrix points than does the placement matrix.) If the board contains parts on both sides of the board, then a matrix must exist on each side of the board.

The user should define a matrix small enough where only 1 or 2 components exist within a matrix block (not ON the matrix, in it). The command works from the matrix origin, in the sequence specified by the user. The number of the rows and columns are not important; the number of components that fall within the matrix block is. When more than one component falls within a block, the components are sorted by the component origin in the sequence the user has specified. If there is no matrix selected, or there is more than one matrix selected for any side of the board, then the command will not run; and an appropriate error message is generated.

Should, after AUTO RENAME, you find that the Renaming resulted in a part, or part being renamed out of sequence, most likely a part origin has been placed on a row or column, rather than inside the matrix.

Upon entering the command, the following prompt will appear:

Renumber Horizontally or Vertically (H/V)? [H]:

If "H" is entered, then the parts are renumbered in rows; otherwise, they are renumbered by columns.

The parts may be renumbered starting at either the left, or right edge of the board going in ascending order to the opposite edge. This is selected with the following prompt:

Renumber Left to Right (L/R)? [L]:

The parts may also be renumbered starting at either the top or bottom edge of the board going in ascending order to the opposite edge. This is selected with the following prompt:

Renumber Top to Bottom (T/B)? [T]:

Having selected the order and direction in which the parts are to be renumbered, the following prompt will appear:

Ref-Des Prefixes to be Renumbered? [ALL]:

To renumber ALL parts, use <CR> in response to this prompt. The family of parts to be renumbered (if not all) is selected by specifying the alphabetic prefix. Several prefixes may be listed separated by commas, or spaces, i.e., U,R,C,CR,Q.

Having selected the order for Renaming, the following prompt would appear:

OUTPUT SCHEMATIC ECO REPORT FILE NAME (CR = PRINTER):

Respond with the name of the file to be created, which in turn, will be used to update the PADS-LOGIC Schematic.

After the output schematic ECO File has been named, the parts will be renumbered; and the board will be re-displayed with the new reference designators.

NOTE: Before Renaming, make an extra JOB SAVE to use, in the event you are not satisfied with the results and wish to start afresh.

• Grid - Set the Grid to the appropriate grid for track routing. Generally, it should be set to the center to center spacing you want to have between adjacent tracks. A Grid of less than .010" is not recommended, unless you're trying to squeeze by some object and need to go off the normal grid used for routing the board.
• Level indicates the current level or layer that you are working on. As you change layers, this value will change.
• The layer pairs command shows the currently selected pair of layers. This is used in two ways: during interactive routing on one of these layers, selecting Change Level automatically switched to the other member of the pair; the two layers listed are the routing layers for the standard autorouter (PL 1 2)
• The V information gives the currently selected via type: T (Thru Hole), M (Micro) and B (Buried), that will be inserted when you change layers.

In addition, at any time while a connection or track has been picked, a message is given identifying the to-from information of the picked connection or track, i.e, R1.1 U6.2, together with the Signal Name and Track Width.

7.1 Interactive Routing

The Interactive Routing commands are as follows:

1. The RUT CONN command is used to convert a rubberband connection to a complete or partially completed track.

2. The MODIFY command is used to change the path of a previously routed or a partially routed track.

3. The UNROUTE command is used to convert a routed or partially routed track back to a rubberband connection.

4. The SEL TEAR command is used to enable or disable the tear drop pad function on individual track segments.

5. LEN OF allows the measuring of the entire Net, a single connection (route) or a segment of a route.

7.2 Converting a Connection to a Track

To route a connection, place the cursor over the rubberband connection to be routed and selecting RUT CONN.

When the connection is picked, the Information Window provides a three-line message, indicating the Signal Name of the connection, the To-From information, i.e., U2.12 U1.10, and the Track Width. In addition, the Menu Window changes to that of Figure 7.2.

1 Add Crn 2 Del Crn 3 Angle 4 Level 5
6 7 Fin Net 8 End 9 Fin Pin 10 Exit

Figure 7.2 - ROUTING OPTIONS

When selecting a connection to be routed, place the cursor over the connection near one of the pads associated with the connection. The nearest pad will be considered the "beginning pad." The track will proceed out from the beginning pad toward the cursor on the Default Level indicated.

The track originates on a Level or Layer according to the following:

- If the component pads exist on the selected layers, then the initial and final track segment is set at the Default Level setting. This is modified by the L modeless command or the Level command.
- The Default Level, upon startup, is Layer 1. After routing a track, the Default Level is the layer at the time of completion of the proceeding track.
- If the pad from which the track exits does not have a padstack on the current layer, but does have it on the other member of the Layer Pair, the track will start on the other layer in the Layer Pair.
- If the pad from which the track exits does not have a padstack on the current layer or on the other member of the Layer Pair, the track will start on the next layer for which a padstack is defined.

The track segment will follow the cursor on either of the four 90° X/Y directions or the four 45° directions, simply by pulling the cursor near the direction desired. This command allows Vertical, Horizontal, or 45° Diagonal. (ANGLE adds the capability of routing at Any Angle.)

That remaining section of the connection between the end of the routed track and the Ending pad follows the cursor, and its length/direction always changes as the track is in process of being routed. This provides a "homing beacon" to help the designer finish at the terminating or Ending pad. (see Figure 7.3 below). The track can be completed to the Ending pad using FIN PIN (F9) or using FIN NET (F7) and can be tied into any pin in the Net or any routed track segment, within the Net.

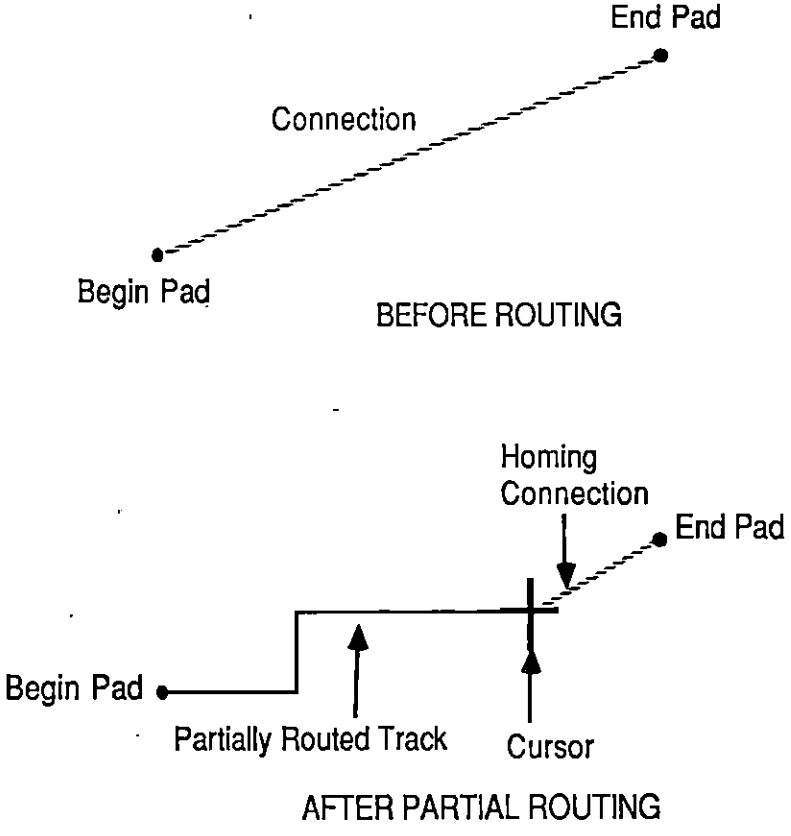


Figure 7.3 - TRACK WITH HOMING CONNECTION

Highlight is an aid to interactive routing in that when "Highlight" is ON, all the connections and pads in the entire net are Highlighted with the "complimentary pair" color used to set up the color for tracks. To

turn "Highlight" ON, use the modeless command of H <CR>. To turn "Highlight" OFF, repeat H <CR>. A prompt message provides status. Alternatively, within the SETUP, PARAMS menu, the user may select CURRENT NET HIGHLIGHT to turn the highlighting on or off.

The user may change the Current Track Width at any time, for the current segment and subsequent segments in the track, by using the WIDTH modeless command, I.E. WX <CR>, where X is the new width. REAL WIDTH must be set to show width of track (SETUP, PARAMS, or Modeless command Rxx). Track segments that are already defined will not change width. This function is useful for track necking.

7.2.1 Adding Corners While Routing

While routing the connection, it is necessary to include corners in the track to change direction. The selection of ADD CORNER will insert a corner at the current cursor location. The track can continue from the corner in any of four Orthogonal headings or at any 45° direction, determined by the direction of the cursor. Adding corners is also used at the start of a necked track, as well as the end of the necked track section.

7.2.2 Deleting a Corner While Routing

It may be necessary to "back up" and try a different path. You can back up to the previous corner in the track with DEL CORNER. This deletes previously added corner, and may be done multiple times.

7.2.3 Making Any Angle Corner in Routes

If a track corner angle other than 90° or 45° is desired, select ANGLE, after the corner is added with ADD CORNER. This allows the line to be drawn away from the corner at any angle depending upon the Default Grid setting and the length of the line. Once a corner is inserted, the next segment defaults back to 45° and 90° snap grids. If the user wishes to continue using odd angles, ANGLE must be selected again.

The ANGLE command acts as a toggle switch; selecting ANGLE again will force the track segment back to either Orthogonal or Diagonal.

7.2.4 Changing Layers in a Route

The LEVEL (F4) command is used to change the Level on which the track is entered, by inserting a via or feedthru at the last corner on the track.

If the track is currently on one of the layers specified in the Layer Pair display, then when LEVEL is selected, the current track segment is automatically switched to the opposite pair in the Paired Layer setting, and a via of the current via type is inserted at the last track corner. If no corner has been put in the track, the level or layer will be changed at the Beginning Pad. If the Pad is a surface pad that has a pad stack only on the current layer, a warning message appears in the Prompt Line and the command is ignored.

If the current layer is not one of the layers specified in the Layer Pair command, the first layer in the layer pair is automatically selected.

The user may also change the Current Level at any time, adding a via at the last corner by using the L (Level) modeless command. See Section 1.4 for details.

7.2.5 Ending the Track

The END command is used to stop the routing of a track without finishing on the ending pad. When the command is selected, the track is stopped at its current position, a via pad is inserted and a connection will appear connecting the end of the track with the end pad. During subsequent routing, this unrouted connection is treated as an unrouted connection. The example below illustrates the implementation of END.

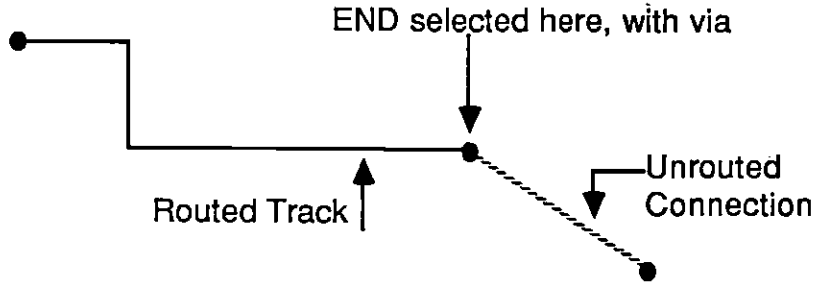


Figure 7.4 - IMPLEMENTATION OF END

7.2.6 Finishing the Route (FIN PIN)

This option is used to complete the conversion of a connection to a track. The action is at the terminating or End pad of the connection. When FIN PIN (F9) is selected, the track will be completed at the End pad along one of two paths:

1. If the track segment is directly to the End pad, it will terminate at the pad.
2. If the final track segment does not intercept the End pad, then the track will proceed to the nearest point to the End pad, make a right angle corner toward the pad and finish at the pad.

The track remains on the current layer unless the padstack for the ending pad is not defined on the current layer. If this happens, the final track segment changes level to insure that the segment is on a valid layer, and a via is inserted automatically.

No account is taken of tracks, vias, or pads that may be shorted while completing the last track segment. For this reason, FIN PIN should be used only when near the Terminating or End pad, where the results of the FIN PIN action the path will take can be predicted with accuracy.

7.2.7 Aborting a Track Route

Selection of the EXIT command reverts the track back to its connection state before the routing option was started.

7.2.8 T-Routing

T-Routing permits the start of a track from any pin or any track in a Net and the completion of the track at any pin or any track within the Net. The feature FIN NET (F7) makes this possible.

To start a track on a track or net in the Net, place the cursor over the track and select RUT CONN. A segment track will be initiated or "T'd" to the routed track. To finish the track on another track or pin within the same Net, place the cursor over the track or pin and select FIN NET (F7). The track being routed will be "T'd" into the previously routed track. If necessary, a via will be placed at the "T" to connect the two tracks.

If a "loop" is created while T-Routing between tracks, a prompt message:

Delete loop in Net (Y/N)

permits the deletion or retention of the loop.

HINT: T-Routing is an excellent aid for adding additional isolation Ground tracks to the Ground Net, without adding Ground connections.

FIN PIN (F9) finishes only on the pin at the end of the connection. To finish on any pin in the Net, place the cursor over the desired pin and select FIN NET (F7).

HINT: FIN NET (F7) is most useful for "reordering" or resequencing the pins within a Net.

T-Routing provides a very easy method by which an existing track from Pin A to Pin B can be rerouted. Simply place the cursor over one of the pins. Select RUT CONN (F1), route the new path to Pin B and select FIN NET (F7). The old track will be automatically erased, leaving the new track.

7.2.8.1 T-Routing Features

The T-Routing features provide an alternative to the Interactive routing of a connection as discussed in Section 7.2 (Converting a Track to a Connection). With T-Routing, simply place the cursor over the beginning pin of a connection and select RUT CONN (F1). The track will start from the beginning pin and the connection will be highlighted, but will remain in place during the routing. The same interactive routing commands will be available as in Section 7.2. Use ADD CRN (F1), LEVEL (F4), etc. to route the track to the vicinity of the terminating pin, and use FIN NET (F7).

7.3 Modifying Routes

It is often necessary to modify tracks that have been created by the autorouter. The MODIFY command is used to change the track paths of a previously routed or partially routed track. The command supports the movement of track segments or corners, and the changing of layer, via type, or track widths. When the command is selected, the Menu window changes to the MODIFY options as shown in Figure 7.5 below:

1 Move Crn 2 Move Seg 3 Cut Seg 4 Add Crn 5 Delete Crn
6 Via Type 7 Line Wid 8 Level 9 Unroute 10 Exit

Figure 7.5 - TRACK MODIFY OPTIONS

A track is modified by placing the cursor over the section or corner of the track to be modified and then selecting the appropriate menu option command. When the track is picked, the Information Window (if on) will indicate the name of the Track being modified, the to-from information for the connection, and finally, the Track Width.

In general, the commands work similarly to the 2-D line, Copper, and Board Outline Modify commands. For more information about general operations of MOVE CORNER, MOVE SEGMENT, refer to Section 5.3.4.

NEW LEVEL [X]>

If the change of Level brings the segment of the track to the same Level as the other segments of the track to which it is joined, vias that were in the track at the end of the segment will be automatically removed. If going from the same level to another level, feedthrus are put in automatically of the type indicated in the Via setting of the Default table.

NOTE: PADS-2000 does not check for spacing violations during the CHANGE LEVEL activities, therefore, DRC spacing violations may be created with this command. These will detected in DRC Check.

7.3.9 Converting Routes Back to Connections

The UNROUTE (F9) command is used to undo a completely routed track or partially routed track and convert it from the track condition back to its original rubberband connection condition. This command is generally used when it is more convenient to completely reroute a track rather than modify it or when you want to convert an area of the board back to connections for rerouting.

To implement, place the cursor over the track to be unrouted and press Function Key F7. The track will disappear from the CRT and the message:

CONFIRM UNROUTING Y/(N)?

will appear and in response, you either confirm (Y) or deny (N) the Unrouting activity. ABORT, cancels the Unroute activity and reverts the status back to the Route Editor.

7.4 Autorouting with PADS-2000

The PADS-2000 Autorouter contains three separate autorouters, plus a Push and Shove Autorouter. One is specifically for making Power and Ground connections; one is optimized for Memory style connections, and one is a general purpose Maze router. Judicious use of these autorouters can improve routing time in a design. All routers are fully integrated with the PADS-2000 package and can be used in conjunction with the Interactive router. This section consists of five sections: an

introduction describing how to get ready to run the autorouters plus a detailed section on each of the routers.

7.5.1 Selecting Grid, Layer Pair and Via Type for Routing

Before starting the autorouter you must be sure that all parameters are configured correctly. Prior to running the autorouter you must set the Routing Grid, select the Layer Pair, select the Via Type, and turn on the type of connections you intend to route. Use the modeless commands to establish the routing grid size, routing layers, and via type. Refer to the Section 1.4 for details on using the modeless commands.

The PL Layer Pair command defines the layers to be routed. The first layer called out in the modeless command is the Horizontal track layer and the second layer is the Vertical layer. All autorouters are capable of placing short "wrong-way" segments (for example, vertical segments on the horizontal layer).

The Via Type is used by the maze router only. The other routers never insert vias. In the example above, the VT instructs the maze router to use normal "through" vias. It can also use buried or blind vias.

NOTE: The Components and the Connections to be routed MUST be enclosed by a Board Outline. Without a Board Outline, the router will attempt to route over the entire 56.00" x 56.00" of the system maximum area; in the process error messages will interrupt the routing process.

7.5.2 Selecting Visible Nets

The routers will only attempt to route visible or displayed connections in the design. You must set the Net Attribute display parameter before running the autorouters to insure that all connections that are to be routed are displayed visible.

For the Power and Ground router, turn ON the display of power and ground nets and turn OFF the display of all other nets. This allows the autorouter to access power and ground only.

On a two-layer board complete all the Power and Ground routes automatically and/or manually before starting the Memory or Maze

routes. For a multi-layer board with Power and Ground planes (routing not required), you must turn OFF the display of Power and Ground signals before using the Memory or Maze routers.

7.5.3 Selecting the Routing Grid

Select the autorouter by selecting the the "AutoRoute" menu (F2 key).

You will be asked to define the routing grid. The default routing grid size is remembered from the last Route or from the default Grid setting. A message appears as follows:

Routing Grid [25]>

If the grid is correct, select <cr>. Otherwise, enter the new routing grid size, either 10, 12.5, 16.67, 20, 25, or 50, followed by <cr>. All tracks and track corners will be placed on this grid.

You will then be asked for the Via grid with the prompt:

Via Grid [0]>

Enter the via grid size, followed by <cr>. The via grid should be a multiple of the routing grid or may equal the routing grid. The Maze router will only attempt to locate vias on via grid intersections. Pre-existing vias will be accepted on any grid but will only be shared if they lie on the via grid.

7.5.4 Selecting Individual Router Passes

The "router passes" control which of the autorouters will be used. You are invited to select the Router Pass with the message:

Use Set-Up (F7) to select the Routing Pass

A "Pop-Up" Window as follows is presented:

No Via/Dip
Short P/G
Long P/G

Short Memory
Long Memory
Horizontal/Vertical
Short Only
Long Channel
One Via
Two Vias
Three Vias
Four Vias
Five Vias
Six Vias
Eight Vias
Any Way
Via Reduce
Space Check

Select the desired routing pass or multiple passes with the cursor. It will change color. If more than one pass is selected, the router will progress through each of the selected passes in the order in which they are listed. Always select from the top of the pop-up window to the bottom of the window. That is Route Power and Ground first, followed by Memory, Short Only, etc. In fact, follow the practice of Routing Power and Ground, then stopping and interactively routing any power and ground connections not auto routed. Then reenter the auto router for Memory, Short, etc. After routing passes are selected, exit using EXIT.

The NO VIA/DIP selection will NOT place vias under IC components if (1) NO VIA/DIP is selected and (2) the IC has been identified in the Library Part Type as TTL, CMOS, etc.

Short P/G and Long P/G are used for routing power and ground tracks on two layer boards. They will route the tracks either vertically under ICs or horizontally across the ICs. They work well on digital designs with ICs placed on a regular matrix. To be selected for routing, a connection must be exactly Vertical or Horizontal. A Diagonal connection will NOT be autorouted. The power and ground connections will be routed with no vias. Short P/G has a maximum connection length of 2" and a maximum wrong way distance of 0.1"(Wrong way is for example vertical tracks on the layer designated to be horizontal). Long P/G has a maximum connection length of 5" and a maximum wrong way distance of .05"

The Memory router routes the memory type connection where the distance between memory IC's pins (measured from pin 1 to pin 1 of IC's), for short memory is up to 0.7" and long memory is up to 1.5". Memory connections are those that are exactly horizontal or vertical.

The Maze router supports up to 10 passes starting with the "Short" pass. The pitch between component pads must be at least .100". The Horizontal/Vertical Router routes short connections that are near Horizontal or near Vertical.

The pass names, i.e., "One Via" etc., are intended to represent typical routes that would be produced by the pass but are not exact. For example, the "Four Vias" pass would use up to four vias but might install a route with five vias if it could do so with no "wrong-ways".

For a quick way to route the easy connections, use the first three or four passes. For a medium elapsed time run with reasonable completion, select every other pass. For maximum completion on an overnight run (where time is irrelevant), select all passes.

The "Any Way" pass routes with an unlimited number of vias and corners. If all the connections are not autorouted, it may make the board difficult to complete manually since it searches all possible vacant grids on the board. Nonetheless, going immediately to "any way" only, without the proceeding passes (except Power, Ground and Memory) may route more connections in a shorter period of time than using the progressively more different pass method, although the total number of routed tracks may be less.

The last two passes are not routing passes. One is a Via reduction pass, the other is a Space Check. The via reduction pass will swap segments of a route to the other layer when doing so will safely eliminate a via. This pass can be used as a clean-up following auto routing. It can also be used independently of the maze router after manually routing.

The space check pass is provided as a convenience to allow checking as part of the routing pass, such as at the end of an overnight routing pass.

When finished selecting Passes, Exit with EXIT (F10).

7.5.5 Selecting Connections to be Routed

The next step is to select the connections to be routed. You can select the connections individually (F1), by window (F2), by signal name (F3), or all unrouted connections on the board (F4).

To select connections individually, place the cursor over the rubberband connection to be routed, then press SELECT (F1). If the connection meets the router's rules (see specific router descriptions for details), the rubberband will be erased,and replaced with the routed track. If the connection does not meet the router's rules, a message will be displayed indicating why the connection was rejected. To select another connection, move the cursor and press "Select" again.

To route those connections that originate and end within the display window of the CRT, select WINDOW (F2). This will route only the connections that are completely contained within the window area.

To route all the connections of a named net select SIGNAL NAME (F3), and respond with the net name and <cr>.

To select all connections on the board, select BOARD (F4). (Recommended)

After the connections category is selected, the unrouted connections are counted and the number of unrouted connections displayed in the Information Window.

After the connections are counted, the system will then write a Security Dump to disk and start the autorouter.

During autorouting the Information window displays the number of connections routed and number of connections not yet routed, together with the name of the current pass. The "Working" line displays pass number percent remaining of that pass. Each pass counts as 100, e.g. 650 means half-way through the current pass and 6 more passes to go. After each pass the total number of routes completed is written to the bottom line of the screen .

All routers dynamically display completed routes. Each new route is updated in the graphics work area as it is completed. You can interrupt any router while it is running by selecting EXIT (F10). If you do, all

routes already completed will be retained in the database, but no further routing will be done. You can then continue working with the board as it existed at the termination or you can read back the security dump made before routing was started.

All routers are restartable. You can freely mix manual and automatic routing. If obstructions prevented routing, you can move the obstruction and then run the autorouter again.

7.5.6 Learning to Use the Autorouter

To appreciate the AutoRouter better and maximize its performance, you should be aware of the following:

The AutoRouter (or any autorouter) is not a substitute for an experienced designer. You will almost always need to interactively complete all the connections as well as interactively route those tracks that have special requirements as to their location. Auto Route on the other hand, used properly, is a great time saver and typically can reduce the time a designer would spend routing the board by 50 to 75%.

The PADS-AutoRouter is very sensitive to the decisions you made in setting up the database for the board, especially placement, pad size, track size, Via size, track widths and spacing. During set-up, the autorouter for a routing pass, Routing Grid, Via Grid, layers, and the selection type of routing will greatly affect the results. These various items are discussed as follows:

Placement

Good placement of components is essential to the success of the router. Always follow the Placement Check List in the Chapter 6. Pay particular attention to minimizing Diagonal Connections (vertical and horizontal) connections - make even the short run as horizontal or vertical as possible.

Connection Hot Spots

Minimize (and eliminate where possible) very dense congestions of rubberband connections on this board. Both the designer and the

Autorouter will encounter difficulty if there is not enough space to route.

Blocking Routing Channels

Critically examine the pad pattern to insure that pads are aligned in such a manner as to not block routing channels. For example, don't allow the Pads of an IC to be staggered .050"with respect to an adjacent IC and thus block each other - put the pads of both IC's on the same routing channel thus this leaving the other channels free for routing.

Pad Sizes, Track Sizes, Spacing and Routing Grid

The ability of the router to place tracks between IC Pads depends entirely on the selection of pad sizes, track size, spacing and routing grid. This combination of variables is perhaps one of the more frequent reasons for poor routing results.

So in all cases, you must do your "numbers" to insure that you are not asking the impossible!. Consider the following:

.070" IC Pad
.015" Track
.010 Space
.025" Routing Grid

Will this combination allow one track between IC Pad?.

With a Routing Grid of .025", there are three routing grids between IC Pads, one .025" from each IC Pad center, and a third at .025", or halfway between the pads. The two routing grids nearest the IC Pads are too close to the IC Pads to route. How about the routing grid at the mid point between the Pads, i.e., .050" from the IC Pad center. The sum of 1/2 IC Pad diameter (.070") + Minimum Space (.010") + 1/2 Track (.008") equals .053". Since 53 is greater than 50, we see from the above that the combination cannot be routed and thus the router would always fail to route. 60 mil IC pads would permit routing. The Lesson: Always do your sums.

Density

The following is an approximate guide to what you should expect in connection completion percentage as a function of board density, if all parameters are selected correctly.

The board density in terms of board area (sq. in) per equivalent 14 Pin IC is perhaps the most significant factor in routing. (You can obtain the board density by using the menu REPORTS and printing the command STATISTICS).

2 Layer Board	Routing Percentage
1 sq. in./IC	95 - 100%
.7 to 1 sq. in./IC	85 - 95%
.6 to .7 sq. in./IC	65 - 85%
Below .6	Suggests 1 Layer

4 Layer - (2 Track - 2 plane)	Routing Percentage
.5 - .7	85%
.4 - .6	70%

There are many variables that affect the routing success and you need to develop a working knowledge of these variables and their results.

We suggest the first approach is to use a board of about 50-100 IC's and start off routing as a 2 layer board, 1 track between IC Pads. Do not select "board" but instead take fairly small window areas and make different routing passes on these windows using different parameter values for pass. By taking a small window within the board you will be able to make a larger number of passes with different parameters, in a given period of time.

By varying the parameters (IC track size, spacing, pad size, routing grid, via grid) on each pass and making quick passes using only "window" you will rapidly acquire a working knowledge of the Auto Router characteristics which will stand you in good stead in routing complete boards.

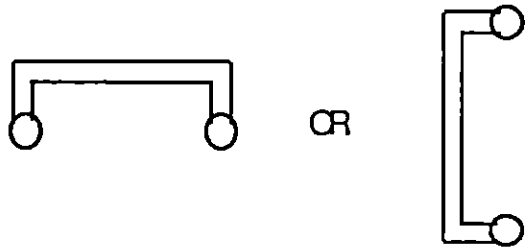
7.6. The Power and Ground Router

The Power-and-Ground Router is designed to automatically route a large percentage of the power and ground connections on a typical two-layer board.

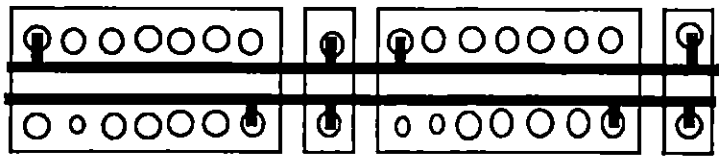
7.6.1 Theory of Operation

Being Heuristic in nature, the Power and Ground router follows pre-determined paths during routing. These pre-determined paths are based upon the assumption that IC's are located in accordance with well established criteria for IC placement. Specifically the assumption is made that IC's and their Decoupling Caps are located along an axis either Vertically or Horizontally on the board. All IC's should be oriented in the same direction and spaced equally apart.

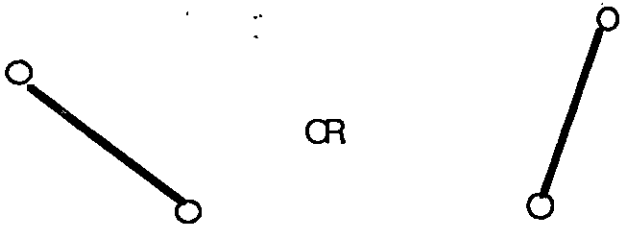
The Power and Ground Router always expects to make a three-segment track for a connection as pictured below:



Where the middle segment runs parallel to the long axis of the IC, the Power and Ground Router would route a row of IC's as pictured.



If the Router can not use its three segment Heuristic pattern, it will not complete even the simplest of connections. That is why you will see connection patterns that are a direct path, such as:



will not route since they are not either exactly Vertical or Horizontal and can not be completed with the three segment Heuristic pattern. This means that the Power and Ground Router will do an excellent job on the Horizontal and Vertical connections of Digital Boards but should NOT be used on Analog Boards.

7.6.2 Options Supported

Routing grid size is selected by the modeless G command (for example, G25) or in response to the grid question when starting the router.

Space to leave between routes and pads by the Auto Router are taken from the database (see Vol II, section 4.7).

7.6.3 Power-and Ground Router Limitations

Although the Power and Ground Router can be used on signal connections, it will not normally produce the optimal path. Further, the memory and maze routers do not recognize power and ground as such. They depend upon the user to select only power and ground connections using the Net Attribute Display and the names used for the Power and Ground Nets.

The Power and Ground Router uses the point-to-point connections defined at the time that it is run. You must run a Power and Ground style reconnect (vertical or horizontal) prior to running the Power and Ground Router. You can also manually adjust the Power and Ground Cross-Tie connections using the ECO add-connection and delete-

connections (see Vol II, chapter (9). If you present the router with clean busing, you will get better routing.

The Power and Ground router chooses the route path according to the location of the end-points, not according to the board layout. It considers only pads when selecting the path. It ignores copper, text, and pre-existing routes.

Do not expect the Power and Ground Router to route 100% of the Power and Ground connections. The Vertical and Horizontal connections will all generally be routed. Connections that are on the diagonal will not be routed even though there is, to the eye, an obvious and easy path for the track. Again the Router is performing normally and you should not be disturbed at this situation. This is one of the "trade-offs" between speed and performance associated with a Heuristic Router.

Due to the above, do not use the Power and Ground Router on an Analog Board.

7.6.4 Power and Ground Router Check List

(1) Check the track width of Power and Ground connections, pad sizes and spacing, so as to insure the assignments permit routing.

(2) Using NET ATTR (F5) of SETUP (F2), turn Power and Ground ON and turn All OFF. Set LMTYP to either Vertical or Horizontal (Vertical IC's use Vertical LMTYP; Horizontal IC's use Horizontal LMTYP. The Power and Ground Router will then route only the Power and Ground connections.

(3) Run a Length Minimization (Horizontally or Vertically depending upon the orientation of the IC's) for Power and Ground.

(4) Make a Job Save and enter into your Save log.

(5) Select ROUTE, AUTOROUTE, then Short and Long PWR/GND.

(6) Set routing grid = 25.

(7) Select the group of connections you wish to route from:

(F1) SELECT
(F2) WINDOW
(F3) NET
(F4) BOARD

In most instances you will wish to route the entire board. You may, however, wish to check your setup by routing only one connection using SELECT (F1), and choosing the connection with the cursor (incidentally, selecting one connection at a time is an excellent way of determining why the router will not route a particular connection; if you select a connection to be routed and it can not be routed, a message will tell you why). You may want to use this technique when you are learning to use the router, but not for real Board Design..

(8) Select BOARD (F4) to route the entire board. The Information window will indicate the number of Power and Ground connections that have been selected for routing.

(9) The router will start routing and the results are displayed in the Information window as follows:

TO GO XX — where XX are the remaining connections to be routed

ROUTED YY — those connections that have been routed

POWER/GROUND
WORKING — when Working disappears the routing pass is over and the results are displayed in the RED prompt line.

(10) At this time, on large complex boards where there are a number of Power and Ground connections left unrouted, you might run a Length Minimization opposite to the previous Length Minimization (if you had previously run the Length Minimization as Vertical, then run this pass as Horizontal).

(11) Make another routing pass using Power and Ground Router.

(12) At this point, use the interactive router and finish routing all the Power and Ground connections that remain after the last routing pass. Do not hesitate to tidy up any of the tracks put in by the Autorouter.

(13) Make a Job Save and enter into the Job Save log.

7.7 The Memory Router

The memory router is designed to automatically route typical memory connections using five-segment routes, on a single layer.

7.7.1 Theory of Operation

The Memory router is a Heuristic Router. That is, it selects the memory type connection, then makes the routing path based on the orientation of the components. Being Heuristic, the Router follows the typical path for routing a memory track and does not search for alternate paths.

Each "memory like" connection is first classified as vertical or horizontal. The first layer of the current layer pair is used for horizontal routes between vertically oriented DIPs. The second layer of the current layer pair is used for vertical routes between horizontally oriented DIPs. If the connection is not exactly vertical or horizontal, it is rejected and no action is taken.

Only five segment routes are constructed. As a special case a direct point-to-point single segment track will be routed where appropriate. Connections from the left side of an IC to the left side of the next IC go half-way across the IC, then up at a 45 degree angle, then half-way to the next IC, then down at a 45 degree angle, and finally on to the end pad. Conversely, connections from the right side of an IC to the right side of the next IC go under the intervening pad. Similar rules apply to horizontally oriented ICs.

7.7.2 Options Supported

The routing grid is selected by the modeless G command (for example G25) or in response to the grid question when selecting the memory router. Grid selection is critical to the path selected by the router. For normal memory using one track between path technology, a grid of 25 is essential.

11. You may wish to make a pass at 3 VIAS and 5 VIAS, STOP and use the Interactive Router to modify any Automatic Router placed tracks that tend to block other tracks (such as around the I/O Connector), then resume Autoroute.

12. At the end of the last Auto Routing Pass, select VIA REDUCE, to remove any unnecessary VIAS.

13. If running an overnight pass, select Space Check so as to result in having the board checked, before starting to work the next morning.

14. After Set-Up, select Connections to be routed from:

1 Select 2 Window 3 Net 4 Board

Selecting a single connection is not particularly useful in the Maze Router. Normally you will want to use either Window (F2) or Board (F4).

7.9 Determining the Length of a Track

The LEN OF command is used to determine the length of a connection, track, track segment, or net. This is useful for high speed or other critical circuit design, where track length can affect signal performance.

When the option is selected, the menu choices are:

1) Net
2) Route
3) Segment
4) Net Name

To select one of the first 3 options, put the cursor on the track or connection to be measured and select the option. If Net or Route is selected, the message displayed is:

Net length of signal $$$ is:
Unrouted: xxx (+) Routed yyyy (=) Total zzzz (mm, inch, mil)

where

$$$ is the signal name
xxx is the length of connection in the net in the units selected
yyy is the routed portion of the net
zzz is the combined connection and route length.

If Segment is selected, the length of the track segment (or connection) is displayed.

If Net Name is selected, the message prompt is displayed:

Enter the Net Name for the Length Calculation >

Type the name of the net to be evaluated. The total length of the connections and routes in the net is displayed at the prompt line, as explained above.

7.10 The Shove Router

The Shove Router is not a batch autorouter. That is to say, it does not run unattended. Instead it should be considered as "interactively controlled". There are three routing options in the Shove Router: Auto Path, Auto Steer and Man Path.

Using AUTO PATH, the designer selects the connections, in the sequence in which the connections are to be autorouted. Upon selection of a connection, the autorouter displays the track pattern the autorouter will produce if the Designer concurs.

If the designer prefers a different track path, the AUTO STEER feature allows the Designer to steer the autorouter by defining a series of points along the path and the Shove autorouter will "push and shove" existing tracks aside to obtain the exact defined path. In this manner, the "steering" feature provides two major advantages: (1) the designer can define the path so as to meet the requirements of the critical tracks on the board, and (2) take advantage of the autorouter to quickly autoroute the track in the desired track pattern. Following the steering or autorouting of a track to a specific pattern, the track can be locked down so as to not be "pushed and shoved" during the routing of other connections.

The Shove router also has an interactive routing feature that allows the designer to manually route critical paths to an even more precise location. During this manual routing an automatic Design Rules Checking feature does not allow spacing errors to be made.

When you enter the Shove Router, PADS-2000 creates a new database for the design to make interactive routing possible. This will take a few seconds to do. You will notice a short time delay each time you change the CRT presentation. This is due to the remapping of the database in the Routing Box or window (discussed later). This wait is decreased with faster CPUs.

7.10.1 The Shove Menu Options

SHOVE is the command for selecting the Shove Router. When SHOVE is selected, PADS-2000 searches for a strategy file with the same name as your design and with the extension .SF. If none is found, you will be prompted for the name of the routing strategy file by:

Enter name of Strategy File?

Type the name of strategy file you wish to use, or <cr> to select the Default strategy file. The file is loaded and the Shove menu illustrated in Figure 7.6 appears.

1 Auto Path	**2 Auto Steer**	**3 Man Path**	**4 Strategy**	**5 Check**
6 Rip Seg	**7 Mov Crn**	**8 Unroute**	**9 Undo**	**10 Exit**

Figure 7.6 - The Shove Router Menu

7.10.2 The Auto Path Router

The AUTO command starts the individual track router. When active, the designer interactively selects the connection to be routed. If the connection or track is marked as Routable in the Net Attr display of the SETUP function, an attempt will be made to route it.

This router will find the best possible path for the selected connection, based on the design rules you have specified in the strategy file. In some cases, the router will move, or "Shove", other tracks out of the way to

make room for your new track. Depending upon whether routing a track Horizontally, or Vertically, the AUTO PATH autorouter will place the track on the correct layers, and if necessary puts in vias. The Shove router will not create a spacing violation with other existing tracks or pads. A routed track will almost always stay within a rectangle defined by a diagonal between the Beginning pin and Terminating pin of the connection.

The Autorouter determines a possible path solution and presents it for review at the CRT. The designer may accept the track path, or cancel it. If the path is CONFIRMED, it is autorouted as displayed in the preview.

NOTE: As the cursor is placed over a pad, the REF DES, Pin Number and a Signal Name are all displayed in the lower half of the Information window, if this is visible.

NOTE: On especially difficult connections the autorouter will try several attempted paths before the final path is found.

7.10.3 The Auto Steer Router

You have noticed that while in AUTO , the Autorouter routes the track using an algorithm that results in the shortest track length and minimum number of vias. While these results are desirable for the majority of tracks on a board, from time to time a circuit will contain connections whose position relative to other tracks and parts are more critical than the advantage of having the tracks routed with the shortest track length. Many Analog circuits are in this category. High speed ECL digital tracks are examples of critical connections. Generally in the situation of critical routes, the Designer will want to establish the track pattern.

NOTE: Wherever possible, always route critical tracks first, at the start of the design.

AUTO STEER is the second autorouter contained in the Shove Router. It is generally used for the routing of critical paths where the designer wishes to control the track pattern i.e., place a corner, yet have the advantage of speed from autorouting. The designer selects the connection or track to be routed, then locates each corner of the track pattern and the router automatically routes the connection while pushing

and shoving other routes aside. If a track is selected, only the segment selected will be rerouted.

The Auto Steer mode is an automatic mode where you can "steer" the router along the path desired for a critical connection to a limited degree and the Shove Router will push and shove other unlocked routed tracks aside to make room for the connection being "steered".

The designer defines the general path that the track should take. The Steer router determines the best way to accomplish the selected path.

SPECIAL NOTE FOR STEERED AUTO ROUTING: On most routes, you may find that the Autorouter "has a mind of its own" and will want to route a different track pattern from the desired pattern. Undoubtedly, the Autoroute algorithm "prefers" a shorter or more direct path than your path and will attempt to readjust the track behind the cursor. To minimize this effort, do the following:

- Always reset the Layer Direction (L1=50, L2=50) in STRATEGY.
- Make a practice of using LOCK CRN (F3) as soon as you make the corner.
- Don't make long segments. Insert a corner as you go, even if you are making a straight segment. This helps you keep control.
- Route your critical tracks at the start of the board design, in order to minimize the "Pushing" of other tracks effort.
- For very tight control, consider using either of the MANUAL routers.

7.10.4 Routing with On-Line DRC

The Auto Path and Auto Steer routers will not create design rule violations. There is also an interactive router function within the Shove router that allows complete design flexibility but will do on-line DRC, insuring no errors are created. This is the MAN PATH command. It is a fully interactive router wherein the designer makes all the decision concerning the track path, including where to position corners and their track layers, and the via types.

To route a track with this command, position the cursor on the connection or track to be routed. If a track is selected, only the segment will be converted to a connection, and the command functions similarly to the Reroute command. Once a route is identified, the track segment attaches is to the cursor and the path can be defined.

The MAN PATH command provides the same interactive routing options as the PADS-2000 interactive router, with the exception that tracks can only be on 90 and 45 degree corners, so the Angle option is not available.

While routing,the router supports on-line Design Rule Checking, which prevents placing of a track too close to other tracks or pads. If the path being defined results at any time in a DRC violation, the track becomes invisible until the violation is removed. Moving the violating segment will cause the track to reappear. Because of this feature, it is impossible to create a DRC error.

7.10.5 Checking with the Shove Router

The CHECK command is used to run a space check. It is identical to the operation of the Check function described in Chapter 8. It is used primarily when you wish to run a spacing check prior to starting the routing of the board in the Shove Router, and thus is useful to identify the errors before beginning the Shove routing.

7.10.6 The Strategy File

This command permits customizing the routing algorithm for the requirements of the board being designed. The Strategy File can be unique to a specific design, or saved and used on other circuits. The menu options are the following:

1 Strat In	**2 Strat Out**	**3 Mod Rules**	**4**	**5 Mod Nets**
6	**7**	**8**	**9**	**10 Exit**

7.10.6.1 STRAT IN

The strategy used by the autorouter while routing a connection on the board is determined by a STRATEGY file that must be in memory at the same time the board is being routed. Shipped with your the Shove Router are a number of different STRATEGY files used for different types of boards.

There is a default strategy file, called STD.SF, which contains the standard strategy settings. If the user wants to use a special strategy file, they may be loaded from disk. If you modify any strategy options, these changes are stored with the job.

NOTE: We recommend that under most two-layer situations, and certainly until you gain considerable experience with the use of STRATEGY, that you utilize the Standard Strategy file.

You may store a strategy setting to disk, and load it into other designs.

A set of Strategy Files are included with the Shove Router program, for use on different circuits. They are as follows:

STD2.SF	2 Layer, Dual In Line, Horizontal Orientation
STD4.SF	4 Layer, Dual In Line, Horizontal Orientation
STD6.SF	6 Layer, Dual In Line, Horizontal Orientation
STD8.SF	8 Layer, Dual In Line, Horizontal Orientation
STD2V.SF	2 Layer, Dual In Line, Vertical Orientation
STD4V.SF	4 Layer, Dual In Line, Vertical Orientation
STD6V.SF	6 Layer, Dual In Line, Vertical Orientation
STD8V.SF	8 Layer, Dual In Line, Vertical Orientation
SMD2.SF	2 Layer, SMD, Horizontal Orientation
SMD4.SF	4 Layer, SMD, Horizontal Orientation
SMD6.SF	6 Layer, SMD, Horizontal Orientation
SMD8.SF	8 Layer, SMD, Horizontal Orientation
SMD2V.SF	2 Layer, SMD, Vertical Orientation
SMD4V.SF	4 Layer, SMD, Vertical Orientation
SMD6V.SF	6 Layer, SMD, Vertical Orientation
SMD8V.SF	8 Layer, SMD, Vertical Orientation

If there is a board in memory, and you wish to replace the STRATEGY file that has been originally loaded with the board, use STRAT IN and respond with the new strategy file and it will replace the current STRATEGY file and will remain with the board data base until changed.

7.10.6.2 STRAT OUT

If, after you have set new Strategy rules, you wish to SAVE the new cost setting as a new STRATEGY file, for subsequent use on a different circuit, select the STRAT OUT command, and save the strategy under a

unique file name. The extension .SF is automatically added to the file name.

7.10.6.3 MOD RULES

If you wish to review or change details of the current STRATEGY file loaded with the board, select MOD RULES . The rules are changed with the strategy setup form. This operates similarly to other menu forms in PADS-2000, e.g. you change an item by selecting it with the mouse. Most of the entries are aqua and toggle each time the mouse is selected. Those in black require a value in the range allowed to be entered.The number in ( ) will indicate the range of values or costs for the item and the number in the black box indicates the value in the current strategy file being used. The options that can be controlled are as follows:

45 Degree Routing Allowed
Off Grid Routing Allowed
Multiple Via Types Allowed
Via Grid
Via Cost
Retry Cost
Route Limit Cost
Max Number of Route Attempts
Route Box Size
Layer Rules
Level Direction Flags

Those that have an aqua entry box have two or more choices which are toggled each time the entry is selected with the mouse. The other entries accept a value which is typed at the keyboard. With this STRATEGY concept review in mind let us address each of the features involved in setting up STRATEGY.

DIRECTION COST
DIRECTION is the cost of a "right way" routing cell as opposed to a "wrong way" routing cell and is used to determine which way the track segments will generally travel on each layer.

"Towards 50" means less directional preference, "Towards 0" or 100 means more directional preference. Directions will usually be set to 25 and 75 (per routing layer pair) during the Auto Path routing of the board in order to set a proper route flow for optimum completion. When using Auto Steer, set the Layers nearer 50, Layer 1 at 50 and Layer 2 at 50, for example. Twenty-five (Vertical) and Seventy-Five (Horizontal) are defaults for Auto Path Routing.

USAGE COST
Usage Cost is used to determine which layer(s) will have overall preference for routing. Lower Cost means the router will try to use that layer for routing. A higher cost means that the router will avoid using that layer unless there are no other alternatives available to finish the routing of a given connection.

For instance, on a multilayer board if you were to set the cost of the outside layers to 50, and the cost of the inner layers to 75, the inner layers would be twice as expensive (or half as desirable from a routing standpoint) as the outer layers, all other things being equal. Fifty is the default in the STANDARD STRATEGY FILE.

7.10.6.4 The Standard Strategy File

The Standard File (STD.SF) in a 2 layer strategy file is set as follows:

```
*LAYER  1       ENABLED
*LAYER  1       DIRECTION   75
*LAYER  1       COST  50
*LAYER  2       ENABLED
*LAYER  2       DIRECTION   25
*LAYER  2       COST  50
*VIA GRID       0
*VIA COST       50
*RETRY COST     50
*ROUTE LIMIT    85
*ROUTE ATTEMPTS     3
```

```
*ALLOW 45          YES
*MULTI VIAS        OFF
*OFFGRID ROUTING   ENABLED
*ROUTE BOX         180          120
*VIA GRID          SAME AS ROUTING GRID
```

7.10.6.5 MOD NETS

MOD NETS gives the user the ability on a selective basis, to change the routability of a route within the Shove Router, rather than the user exiting the Shove router and setting this information in the NET ATTR menu. Within MOD NETS' all five functions, the user is changing the routability of the route. When any changes are made within this menu, the NET ATTR menu is automatically updated with the appropriate information.

When MOD NETS is selected, the user has the option to SELECT (F1) a net with the cursor or use the KEYBOARD (F2) entry, specifying the net name. Once one of the above is selected, the following appears:

1 Routing 2 Shoving 3 Sharing 4 Ripup 5 Reconn
10 Exit

Select ROUTING (F1) and the following prompt appears:

Is routing enabled for '$$$' [Y]?

If routing is not enabled, the net is removed from the display, thus the connection may not be routed.

Select SHOVING (F2) and the following prompt appears:

Is shoving enabled for '$$$' [Y]?

If SHOVING is not enabled, the net will not be shoved.

Select SHARING (F3) and the following prompt appears:

Is copper sharing enabled for '$$$' [Y]?

If SHARING is not enabled, tracks and copper of the same net will not be shared.

Select RIPUP (F4) and the following prompt appears:

Is ripup enabled for '$$$' [Y]?

If RIPUP is not enabled, tracks of the same net will not be ripped up during Routing.

Select RECONN (F5) and the following prompt appears:

Is reconn enabled for '$$$' [Y]?

If RECONN is not enabled, connections will not be minimized during Routing.

7.10.7 Other Commands in the Shove Router

There are four other commands, described here:

RIP SEG is used to unroute a segment of a track and convert it back to a connection.

MOVE CORNER is used to interactively relocate a corner in an unrouted track.

UNROUTE is used to convert a previously routed track back to a connection.

UNDO is used while routing a connection and it functions so as to undo the previous command. It causes the just changed route to convert to its previous position, in case a mistake has been made. Selecting Undo again, undoes the Undo.

7.11 Selecting Tear Drop Pads

The Tear Drop Pad Generation parameter in the SetUp menu is used to select whether or not the general tear drop facility should be on or off. See section 4.4.3 for details. If the On mode is selected, it is possible to selectively turn off individual tear drops with SEL TEAR command.

To turnoff a tear drop, move the cursor on the track segment that has a tear drop you wish to eliminate and select SEL TEAR. The segment is redrawn with the tear drop gone. The command is a toggle, so the tear drop may be recreated by selecting again.

The tear drop status for the segment is maintained with the segment, so if you turn a selected segment tear drop to OFF and then turn the general tear drop function OFF, then turn it ON again, the selected segments will not have tear drops. (They will need to be individually turned on again.)

CHAPTER 8 - CHECKING THE DESIGN

8.1 Objective

The objective of this chapter is to familiarize you with the Design Rule Checking function. The purpose of Design Rule Checking is to check the minimum spacing between conductive surfaces on each layer of the board. In addition to the Design Rule or Space Check function, there are three other checks : Check ASCII, Check FutNet, and Check Plane

8.2 Design Rule Check

Any item in the database that violates the defined Design Rule spacing is identified by a small, solid filled oblong dot error symbol. The error symbol identifying the violation will be displayed in the color selected for errors for the layer where the error exists.

During the Design Rule Checking sequence, the following Design Rules are checked:

Between pad and pad
Between pad and track
Between track and track
Between pad and via
Between via and track
Between text and text, pad, via & copper
Between copper and copper, text, pad, via and track

Tracks, pads, vias, and copper areas are checked with their actual sizes. Poured copper is not checked by the DRC program since it is "correct by construction." The Space Checking command is accurate to 1 micron.

To use the Design Rule Check, decide whether you wish to check the entire board or a portion, and make the display at the CRT showing

either the entire board or a section of the board as desired for checking. If you make the entire board visible at the CRT, the entire board is checked. If you Window into a section of the board, only the section displayed is checked. More time is required for checking the entire board than for checking a windowed portion of the board. The latter is used when one is concerned with only a section of a board, and the results are wanted quickly.

Only items that are visible on a layer will be checked on that layer. Insure that pads, vias, tracks, text and errors are set Visible on the proper layers.

Access the Main Command Menu and select CHECK. The Menu window changes to that of Figure 8.1 shown below:

1 Spacing	**2 ChkASCII**	**3 ChkFut**	**4 ChkSchma**	**5 Tie Plane**
6	**7**	**8**	**9 Erase**	**10 Exit**

Figure 8.1 - CHECK OPTIONS

The selection of SPACING (F1) will result in the part of the board displayed at the CRT and all its associated layers being checked for all of the conditions listed above. During the process, the word "WORKING..." will appear at the CRT, and upon completion, the CRT will be repainted with any detected errors visible in the colors selected for Errors on each active Layer. The number of errors found is displayed in the prompt line.

If you wish to terminate the checking process, use EXIT. The checking process will be interrupted and halted and any errors detected between the time the check was initiated and the time it was halted, will be retained and displayed.

If you wish to erase the Design Rule spacing indications from the display (a spacing violation still exists), use ERASE. The small error symbol representing the design rule violation will be erased from the database. Run a Design Rule Check again to bring back the error display.

To exit from Design Rule check during or after the check, use EXIT.

NOTE: In order to show a large number of errors over the entire board, multiple errors on a track segment are summarized with a single error indication. The total number of errors however, will be displayed in the Prompt window. Be sure to rerun check after clearing all space errors.

NOTE: A maximum of 500 errors can be shown.

8.2.1 Suggested Usage

The following is included as a suggested methodology for using Design Rule Checking. Route the entire board, taking care, of course, to minimize the number of spacing violations during the routing effort. Run a Design Rule check with the entire board displayed at the CRT. Display the entire board to obtain familiarization with the distribution of errors over the board. (Insure that the appropriate items, i.e., Pads, Tracks, etc., have been made visible at the CRT).

Take one section of the board at a time (by windowing into it). Start from one side of the board and correct the errors in that area. When corrected, run a Check of only that portion of the board.

Systematically move over the entire board a section at a time, cleaning up and then checking the area that has been tidied up. When it is believed that all areas of the board have been checked and corrected, run a check of the entire board again and display the entire board, looking carefully for any error indications or residual errors that have been overlooked. To locate errors easily, make all items but Errors invisible.

HINT: Areas of copper that are to be tied into GROUND should be given the name of GND, and a Ground Track should be run thru the area. By naming the copper area the same as the Ground Track, Checking will treat the copper area and the Ground Track as equal potential, and will not display an error.

8.3 Check ASCII

The ASCII check is used to compare the original connection list and parts list used to start the board against the current data base of the

board, to insure the data base matches the original ASCII file, or if there are differences, are they understood.

To use, select (F2) CHK ASCII. In response to the message:

ASCII INPUT FILE NAME:

Enter the name of the ASCII file that is to be used to check against the board data base in memory.

You will be asked whether to list the differences on your system printer or to disk. A comparison is performed showing any mismatched connections that exist between the current design data base and the ASCII file. Differences are printed out on the printer or sent to a file as potential errors. You should examine the print out or contents of the file to determine the cause of the mismatch and take corrective action if necessary. (See Chapter 13 for Error Messages).

8.4 Check Future Net

This command is used to check the original Future Net in file against the data base.

To use select (F3) CHK FUT and respond to the message:

FUTURE NET INPUT FILE?

Refer to Section 3.7 for additional information. The input file will be compared against the database and any differences printed out. These differences represent potential errors and should be examined with care.

8.5 Check Schema

This command is used to check the original Schema in file against the data base.

To use select (F3) CHK SCHMA and respond to the message:

SCHEMA INPUT FILE?

Refer to Sections 3.10 and 3.7 for additional information. The input file will be compared against the database and any differences printed out. These differences represent potential errors and should be examined with care.

8.6 Check Plane

This command will cause a design check to be run to check that all pins which are tied to nets that are power plane layers are correctly tied. In many CAD systems, it is very easy to forget to tie a ground pin of a component to the plane, resulting in an open circuit. This command insures that the mistake does not happen. It works as follows:

- All nets whose Plane attribute in the NET ATTR command is set to On will be checked.

- All pins that are part of these nets to be checked will be checked.

- A pin is considered tied to the net if it is a pin with through-hole via and a non-zero size pad on all signal layers, or if it is routed to another pin that is tied to the plane, or if it is partially routed to a standard through-hole via. If none of these conditions applies, the pin is considered not connected to the plane.

- Pins that are not connected to the plane will be flagged with an error marker.

CHAPTER 9 - THE ENGINEERING CHANGES (ECO) MENU

This chapter will familiarize the reader with the use of the ECO menu commands to make design changes to an existing design.

The ECO Menu provides eight Engineering Change functions as follows:

- Permits the Adding of Connections.
- Allows the Deletion of Connections.
- Used to Rename a Connection Net.
- Permits the Disconnecting of Connections from a Component Pin.
- Permits the Adding of a Part to the board.
- Allows the Deletion of a Part from the board.
- Permits the Renaming of a Part.
- Allows the Changing of one Part Type for another.
- Allows the addition of routed tracks

An ECO file from the board may also contain Gate and Pin Swapping.

When ECO is selected the Menu changes to:

1 To Sch **2 From Sch**

10 Exit

TO SCH is used to generate a file on the changes performed while in ECO, that can be passed back to the originating Schematic where, in PADS-LOGIC, the change is incorporated either manually or automatically.

FROM SCH is used to take a change file from the originating Schematic in PADS-LOGIC, read the change file into PADS and automatically update the board database.

9.1 Making an ECO in the Board

When TO SCH (F1) is selected, the message:

OUTPUT ECO REPORT FILE NAME: (CR=PRINTER)

appears. Type <CR> if you want a hardcopy at the printer, and a file is not to be created. Give a filename if you wish the changes to go to an ECO file. An ECO file is created in the subdirectory FILES with the extension .ECO. The format for an output ECO File is shown in Appendix A3.

NOTE: The user is strongly advised to go to a file, thus a hardcopy may be printed later.

You may accumulate a number of changes over time and they will be included in the same output ECO file if the same ECO Report File Name is used in all instances. If you use the same file name, you will be asked:

APPEND FILE [\PADS\FILES\X.ECO] Y/(N)?

After the output ECO file is given a name, the Menu changes as follows:

1 Add Conn 2 DelConn 3 Ren Net 4 DisConn 5 Add Part
6 Del Part 7 Ren Part 8 Chg Part 9 Add Rut 10 Exit

Figure 9.1 - ECO SUBROUTINES

The command options are identical to those found in the On-the-Fly menu, with the exception that there is no Mod Track command for editing the paths of existing routes and there are two new commands, CHANGE PART and ADD ROUTE, described here. You do not create an ECO file listing while using On-the-Fly .

9.1.1 Changing a Part Type

The option of Change Part allows the changing of Part Types on the board from one type to a different Part Type. Two options are available:

(1) change the Part Type of an individual part, or

(2) change all parts on the board of a particular Part Type.

NOTE: The new Part Type must have at least the same number of pins as the old Part Type.

When CHANGE PART is selected, the menu changes as shown in Fig 9.2.

1 One Part	**2 All Parts**	**3**	**4**	**5**
6	**7**	**8**	**9**	**10 Exit**

Figure 9.2 - CHANGE PART OPTIONS

If it is desired to change an individual part on the board select ONE PART (F1). The menu changes to allow the choice of the individual part by either selecting the component to change with the mouse or typing its Reference name with the Keyboard. In either instance the following message is displayed:

PART TYPE FOR PART XXX IS YYY.
NEW PART TYPE>

Enter the new part type. If the Part Type is available either on the board or in the System Library, the message:

CONFIRM NEW PART XXX Y/(N)?

is provided. A confirmation will result in the Part Type change, and a confirming message will be given.

If it is desired to change all components on the board of a particular part type, select (F2) ALL PARTS. The message:

NAME OF PART TYPE IN JOB TO REPLACE>

requests the entry of the part to be replaced. After the entry, the message:

NAME OF PART TYPE TO REPLACE PART TYPE XXX WITH>

asks for the replacement part type. The entry of the Part Type and confirmation of the replacement will result in the replacement of the Part Types on the board and a hardcopy message to that effect.

If a Part Type does not have at least the same or greater number of pins than the part being replaced, the replacement will not occur and the following message will be given:

NEW PART TYPE HAS LESS PINS THAN ORIGINAL PART

If the replacement Part Type does not exist either on the board or in the System Part Library, the following message will be output:

PART TYPE CANNOT BE FOUND

9.1.2 Adding a Route

It is possible to add tracks to the database without first adding a connection. (A connection will, of course, be added to the ECO file.) To use ADD RUT, be sure the Width is set to the desired track width, place the cursor over the Starting terminal and select ADD RUT (F9). The track will start from the terminal and then all the options of the ADD ROUTE routine are available. If the user wishes to change the width of the current route, they may enter the modeless command W, followed by the desired width.

9.2. Updating a Design with an ECO from the Schematic

The FROM SCH command is used to read an ECO change order file created in the PADS-Logic logic capture system and automatically update the board with the changes contained in the ECO file.

To use the FROM SCH command:

1) The database of the board to be updated must be in memory and displayed at the CRT.

2) The file containing the ECO changes from the changed Schematic must be in the PADS\FILE directory and named XXX.ECO where XXX is the file name and uses the extension .ECO. The file MUST be in the format as described in Appendix A-3. If not, the file will not be accepted and error messages will result.

3) Select (F2), FROM SCH. In response to the message:

INPUT SCH FILE NAME (CR=ECO.ECO)

Type in the File Name <CR> (The extension is not necessary)

4) The file will be read in and merged with the board database in memory. In the process, the From Schematic ECO file is checked for errors. If there are errors, the error message will result and the offending item will NOT be incorporated as a change in the board database. The database in memory is changed in accordance with the FROM SCH file. All "logical" changes that can be incorporated in a single add/delete/change fashion will be incorporated. These changes that are not "logical" or requiring a two-step action, will not be incorporated and an error message will be outputted. Changes that cannot be made in the board are ADD PART and ADD CONNECTION.

CHAPTER 10 - PRODUCING REPORTS

10.1 Objective

The objective of this chapter is to familiarize you with the REPORTS menu.

This menu is used to provide five different types of information on the board database in memory. These five types of information are:

1. A listing of the Connection Nets
2. Listing of unused items, such as pins on a part
3. Statistical data of the board in memory
4. Limits
5. Part Lists

Reports can be directly outputted to the printer or created as a file to be printed at a later time.

10.2 Reports Menu

To use the Reports menu select REPORT. When selected, the Menu window changes as shown in Figure 10.1 below:

1 Net List	**2**	**3 Unused**	**4 Stats**	**5 Limits**
6 PRTList1	**7 PRTList2**	**8**	**9**	**10 Exit**

Figure 10.1 - REPORTS OPTIONS

To select any of the seven Reports options, select the appropriate Function Key. The Window will change to list the options selected and the following message appears:

XXX OUTPUT FILE NAME <CR=PRINTER>:

XXX is the option selected. If you wish to make a file of the option for printing later, enter the name of the file, followed by <CR>. If an immediate hardcopy printout is desired, press <CR> and the pertinent report will be printed out.

10.3 Net List Report

The Net List Report is a listing of the Nets on the board. Each list contains the Net Signal Name, followed by each of the nodes contained within the Net list. The format is as follows:

SIGNAL NET PIN REPORT

Sig $$$172
U1.1 U2.1 U3.1 U4.1 U5.1
R1.2 U6.1
PWR
U1.14 U2.14 U3.14 U4.14 C1.1
C2.1 C3.1

10.4 Unused Report

The Unused Report provides a listing of various items that are in the board database but have been "unused." The primary purpose of the Unused Report is to provide a secondary means of checking the database content. For example, an examination of the "unused pins portion" of the Unused Report could indicate that certain unused gates supposed to be grounded had been overlooked. There are six different items reported by this option:

UNUSED ITEMS REPORT

Unused Part Decal type drawing items
Unused Part Decal items
Unused package items
Unused part items
Signal names with only one pin
Unused part pins

The format of the Unused Report is as follows:

```
    UNUSED ITEMS REPORT

U1      7400
4   5   6   11   12   13
U3      7404
3   4   10  11
U3      7400
3   4   9   12   13
U5      7400
11  12  13
U6      7404
3   4   6   8    9
```

10.5 Statistics

The Statistics Report provides a summary of a number of items associated with the board at a given time.

The format of the Statistics Report is as shown below:

```
Design Time: 0:00  BOARD STATISTICS REPORT

Part Types:          6
Parts Top Side:     46   Bottom Side:   0    Total:  46
Drilled Pads:      590   Undrilled Pads: 0    Total:  590
Thru Vias:         310   Buried Vias:    0

Signal Nets:       176
Connections Routed: 210  Partially: 0  Unrouted: 200
Total: 410

Routed Connection Length (inches)      X: 4110.11
                                       Y:    4260.25
                                   Total:    8370.36


Unrouted Connection Length (inches)    X: 3220.02
                                       Y:    3906.15
                                   Total:    7126.17
```

Number of Copper Clearance Errors: 0
Number of Routing Layers: 2
Size of Board (squared inches): 9.45
Board Density
(sq.in.board area/equiv. 14-pin components): 0.24

The area of board and Density is an approximation in that the area of the board is in square inches, with the area determined by the maximum length and width of the Board Outline.

The Board Density is determined by dividing the board area by the equivalent 14 pin IC count on the board, where this Equivalency count is determined by dividing the total number of component pads on the board by 14.

10.6 JOB LIMITS

This report is valuable to the designer in that it lists the number of various items on the board against the maximum number of the items the software will permit. On large boards, pay particular attention, using this report to insure that you do not exceed any limits.

In the event you get an error message to the effect "not enough room", you have exceeded the array limits of some feature such as Part Type, Nets, etc. Print out the JOB LIMITS report and compare the "ITEM USED" in your board database against the "MAX NUMBER OF ITEMS" to determine where your board has the problem.

A representative Job Limits report is as follows:

JOB LIMITS REPORT

----------MEMORY USAGE----------

Dos Memory Available ---------- 324K Bytes
Extended Memory Available ---------- 6880K Bytes
Total Memory Available ---------- 7204K Bytes

System Memory Used ---------- 24K Bytes
Program Memory Used ---------- 1592K Bytes
Data Base Memory Used ---------- 6144K Bytes

Display List Memory Used ----------	180K Bytes
Total Memory Used ----------	7940K Bytes
RAM Memory Used ----------	7204K Bytes
Disk Memory Used ----------	736K Bytes

ITEM TYPE	MAX NO# ITEMS*	ITEMS USED
Parts	4080	0
Nets	8160	0
Connections	24480	0
Route Segments	118846	0
Part Types	900	0
Gates	1800	0
Gate Pins	9000	0
Gate Sig Pins	3600	0
Pin Num Names	5400	0
Part Decals	1018	0
Terminals	10200	0
Term. Stacks	10200	0
Pad Stacks	400	0
Level Info	1600	0
Drawings	3750	0
Pieces	7500	0
Corners	30000	0
Arcs	3750	0
Text Strings	4096	0
Text Chars.	16383	0
Matrixes	20	0
Deleted Pnts	1000	0
Spacing Errors	500	0

*Numbers will vary as function of memory assigned to database at time of initialization of PADS-2000.

NOTE: Examination of the MEMORY USAGE portion of the report is especially useful. In the above example, the Memory allocated to the database was 6MB. This allocation, together with the program, system and display list requirements will result in disk paging. On large boards, in this instance, an additional 2-4 MB of RAM Memory would be a good investment.

NOTE: The following is a definition of each limit specified in the JOB LIMITS REPORT generated by PADS-2000. The maximum number is variable, pending how much DOS memory and how much Extended memory is available.

PARTS: Total number of physical parts
NETS: Total number of nets
CONNECTIONS: Total number of pin to pin connections.
ROUTE SEGMENTS: Total number of routed segments.
PART TYPES: Total number of different part types (i.e: 7400, 7404, 1k, etc.)
GATES: Total number of different gates (a 7400 has 4 gates). If the package appears more than once in the job, the number of gates does not change.
GATE PINS: Total number of pins defined in gates. A 7400 has 12 gate pins. If the package appears more than once in the job, the number of gate pins does not change.
GATE SIG PINS: Total number of "sigpin" defined in part type file. If the package appears more than once in the job, the number of signal pins does not change.
PIN NAMES: Total number of pin names which were defined in the part type files. If the package appears more than once in the job, the number of pin names does not change.
PART DECALS: Total number of decals, i.e., DIP14, DIP16, as used by multiple part types (7400, 7404...) This number includes alternate decals as well.
TERMINALS: Total number of pads defined, including standard and buried vias, and any alternate decals.
TERMINAL STACKS: Total number of different pad sizes, including vias and alternate parts.
PAD STACKS: Total number of different pads with different shapes/sizes.
LEVEL INFO: Total number of different levels for pads/shapes.

DRAWINGS: Total number of 2-D Line elements, copper and component outlines. If there are 2 DIP14's on a job, it is 1 drawing. The standardvia and buriedvia are also considered drawings, thus bringing the total to 3.
PIECES: Total number of 2-D Line elements, copper, and component outlines. If a component outline consists of several individual lines, the number would total those elements (resistor: body + leads = 3)
CORNERS: Total number of corners in 2-D Lines, copper and component outlines.
ARCS: Total number of arcs in 2-D Lines and decals.
TEXT STRINGS: Total number of DIFFERENT text strings. This is free text in a drawing.
TEXT CHARACTERS: This is free text in a drawing.
MATRIXES: Total number of matrixes in a job. Only one matrix may be active at a time; user should use Delete Matrix.
DELETED POINTS: Total number of deleted points on a matrix.
SPACING ERRORS: Total number of errors allowed for display on all layers.

10.7 PART LISTS

There are two different formats for the Part Lists on the board. These are selected by PART LIST #1 (F6)or PART LIST #2 (F7).

Parts List #1 Format is as follows:

PARTS LIST REPORT --

Reference Designation	Part Type	Logic Type
U1	7404	TTL
U2	7404	TTL
U3	7404	TTL
U4	7404	TTL

Parts List #2 Format is as follows:

PARTS LIST REPORT --

Part Type	Reference Designation	Description
7404	U1 U2 U3 U4	Hex Inverter

CHAPTER 11 - CAM OUTPUTS

The objective of this chapter is to familiarize you with how to generate the Manufacturing aids using the CAM menu.

11.1 The CAM Outputs

Within the CAM menu there are post processors that generate art work and documentation on:
(1) the matrix printer associated with your system
(2) wet ink plotters (either Hewlett Packard or Houston Instruments)
(3) Photoplotter
(4) Laser printer
(4) N/C Drill output

11.2.1 Matrix Printer Plots

The matrix printer plots are "check plots" for visually checking your design. Although 2:1 plots that are photo reduced to 1:1 may be used for prototype board manufacturing, this depends upon the type matrix printer you have, and the criteria of the board. As a general rule you should not plan to use the matrix printer photo reduced output for manufacturing of boards where the photo reduced artwork accuracy must be better than approximately .007".

11.2.2 Wet Ink Plots

A Houston Instruments or Hewlett Packard wet ink plotter can be used to generate wet ink check plots or 2:1 "artwork" plots with filled tracks & pads, that can be photo reduced to generate artwork. In general, artwork produced by the wet ink plotters and photo reduced should have an accuracy of ± .005", depending upon the plotter and the photo reduction process.

11.2.3 Photoplotter

A file is produced which can be transferred to a disk, to drive a disk drive equipped photoplotter. This file produces 1:1 artwork with .001" or greater accuracy, depending upon the photoplotter. The photoplotters currently supported are Gerber photoplotters and Laser photoplotters, using the Gerber format.

11.2.4 Laser Printer

A file of any CAM output is produced which drives the Laser printer. Resulting plots are accurate to approximately ±.003".

The current formats supported are Postscript and HP Laser Jet II. The user has the ability with the Postscript driver to specify up to four shades of gray color using a Postscript laser printer. The HP Laser Jet printer supports one color only.

11.2.4.1 Postscript Laser Printers

Postscript laser printers support a variety of high quality, proportionally spaced text fonts. The user may take advantage of these fonts by setting up a table of fonts to be used in the file PSFONT.DAT in the main software directory (\PADS). All text lines in the design have both a text height and text line width attribute assigned to them. The Postscript laser printer driver uses the text line width as a "font index" to look up which font to use from the file PSFONT.DAT. If the line width specified matches a font in the file, then that font is used to plot that text string. If there is no match, then the normal built-in font that is used for other printers and plotters is used. An example of PSFONT.DAT is shown below:

```
1 Helvetica
2 Helvetica - Bold
3 Times - Roman
4 Times - Bold
5 Courier
6 Courier - Bold
```

The user may extend the table to include other fonts. The user should refer to his laser printer manual for a complete list of fonts supported.

To assign text to a specific font, the user may either change the font index in the file PSFONT.DAT to match the width used in the design, or he may modify the line width of the text items in the routine SETUP,SIZES to match the font index in PSFONT.DAT.

NOTE: The font indices are simple whole numbers without any decimal points, hence the user must have his units set to "Mils" when assigning text line widths to match up with the font indices.

11.2.5 Drill

A file is produced which can be transferred to a disk to drive an Excellon or other drill machine.

All three output devices provide the following outputs:
- General Plot
- Tracks Artwork
- Silk Screen, Top & Bottom
- Assy Drawing, Top & Bottom
- Drill Drawing
- Solder and SMT Paste Masks
- Ground & Power Planes

11.3 Using the CAM Menu

In general the process of initiating either of the four post processors is the same: a series of menu windows containing questions and answers is displayed, which the designer accepts or modifies depending upon the output desired.

From the "MAIN ROUTINE" command menu, select (F9), CAM. The CAM menu asks the question:

SPECIFY CAM OUTPUT SUBDIRECTORY

This command allows you to set up a series of subdirectories into which you can save a number of plotting files for either batch plotting or

sequential plotting. By naming the subdirectories a different name for different boards, you can readily locate your CAM output files for a particular board, or save the CAM commands for subsequent processing of a particular board. You may also set up the aperture, drill, and other CAM defaults uniquely in each sub-directory , and you may have a different sub-directory for each photoplot vendor you use.

The response to the message can be a sub-directory name consisting from one to eight characters. Usually you would choose a name that is an abbreviation of the job name for the board.

After the response to Subdirectory, the menu changes to that as shown in Figure 11.1 shown below:

1 Direct	**2 Batch**	**3 Defaults**	**4 Verify**	**5**
6	**7**	**8**	**9**	**10 Exit**

Figure 11.1 - CAM Menus

1. Direct is selected if you intend to plot immediately.

2. Batch is selected if you wish to run one or more of the CAM selections previously made.

3. Defaults is selected if you wish to modify certain default parameters for post processing your job; for example, the symbols for your Drill Drawing and the apertures for your Gerber photoplotter (if you have this option).

4. Verify is selected if you wish to verify the data in a Gerber output file that you have previously generated. This option will read the Gerber file and plot the data on your pen plotter, matrix printer or laser printer, as an exact plot of what the Gerber plot will be.

11.4. Outputting Directly to Plotters

When DIRECT is selected, the menu in Figure 11.2 is displayed.

SELECT OUTPUT DEVICE TYPE		
PHOTOPLOTTER		
PEN PLOTTER		
MATRIX PRINTER		
LASER PRINTER		
NC/DRILL		
PROCEED	GO BACK	EXIT

Figure 11.2 - SELECT OUTPUT DEVICE SCREEN

The cursor is used to select the output device for subsequent action. Place the cursor on the device name and use mouse button #1 to select. The selected device is highlighted.

The selection of the output device places you in the post processing stream of the device selected. However, the subsequent commands are virtually identical for each device; the differences will be described where appropriate.

SELECT PLOT QUALITY LEVEL
CHECK PLOT
ARTWORK QUALITY

Figure 11.3 - PEN PLOTTER OPTIONS

Selection of PEN PLOTTER will cause a secondary menu shown in Figure 11.3 to appear in the blank area of the current menu.

Selecting CHECK PLOT will cause all items to be plotted at their true size but Pads and Tracks will not be filled. Selecting ARTWORK QUALITY will produce a plot with solid filled pads and tracks that can be used for photo reduction to produce "Artwork" plots. Artwork Quality plots take at least two to three times longer to plot than Check plots.

To proceed to the next menu, select PROCEED. To go back to the previous CRT screen select GO BACK. To exit the CAM menu, select EXIT. When the PROCEED box is selected, the menu shown in Figure 11.4 will appear.

SELECT PLOT TYPE				
PLOT TYPE	SELECT LEVEL			
GENERAL PLOT	1	2	3	4
ARTWORK PLOT	5	6	7	8
SILK SCREEN-TOP SIDE	9	10	11	12
SILK SCREEN-BOTTOM SIDE	13	14	15	16
ASSY DWG-TOP SIDE	17	18	19	20
ASSY DWG-BOTTOM SIDE	21	22	23	24
DRILL DRAWING	25	26	27	28
SOLDER MASK	29	30		
POWER/GND PLANE				
SMD PASTE MASK				
PROCEED	GO BACK		EXIT	

Figure 11.4 - PLOT TYPE

There are three "Decision Areas" on the Plot Type CRT screen:
(1) The Plot Type selection area
(2) The Select Level area
(3) The Action (Proceed, Go Back, Exit) area

The Plot Type Decision area permits the selection of a different plot type as follows:

(1) General Plot
A general purpose plot with the contents selected by the user. The user is allowed to select more than one electrical level at one time.

(2) Artwork Plot
This plot is to be used for artwork for layers containing tracks or routes. The user may not select more than one electrical level at one time.

(3) Silk Screen Top and Bottom Side
Used for generating a Silk Screen for either the top or bottom layer.

(4) Assembly Drawing Top and Bottom Side
Used for generating an Assembly Drawing or Artwork for either the top side or bottom side as required.

(5) Drill Drawing
Used to generate a Drill Pattern for drilling of boards. Different size drills result in different symbols at the hole locations.

(6) Solder Mask
Results in artwork used to mask pads and line items. Pad sizes for Solder Mask are plotted over size by an amount specified later on the Transformation Options screen.

(7) Power/Ground Plane
Used in combination with the inner layers (usually Layers 2 and 3) to generate Power and Ground planes of multi-layer boards. Note: The power and ground connection trees are used to identify the Power or Ground pins at which to flash the thermal relief pad, thus creating a connection to the Plane. All other pins are as set in the Pad criteria of the Part Decals, normally over size so as to NOT connect the pin to the plane when the Artwork is reverse printed.

(8) SMD Paste Mask
This is used for boards containing Surface Mount Devices. It is very similar to a Solder Mask, except that it only generates a pads mask for the Surface Mount Devices. An SMD Paste Mask can be generated for either the top or bottom layer of the board. The software assumes that

all components that have pads just on one side of the board, and have a zero drill specified, are SMD components. If this includes other components that are not intended for the Paste Mask, then these can be deselected later on the Transformations Options menu.

After the PLOT TYPE is selected, the levels or layers are selected that are to be included in the plot file.

The SELECT LEVEL decision area containing the number for level 1 through 30, is used to designate the levels from which data is extracted and incorporated in the plot type previously selected. While normally, data for a single level is used to generate a plot, this command allows data from multiple layers to be selected and merged if desired.

Levels above the actual number of physical or Electrical layers on the board are considered "Documentation Levels". That is, they can be used to incorporate TEXT, LINES, etc. that are not on the artwork. The number and type of levels that can be selected depends upon the PLOT TYPE selected. The levels that can be selected for each Plot Type are as follows:

General Plot	Up to 4 different levels of any type
Artwork Plot	1 Electrical level and 0-3 Documentation levels
Silkscreen/Top Side	0-3 Documentation levels
Silkscreen/Bottom Side	0-3 Documentation levels
Assy. Dwg./Top Side	0-3 Documentation levels
Assy. Dwg./Bottom Side	0-3 Documentation levels
Drill Drawing	1 Electrical level and 0-3 Documentation levels
Solder Mask	1 Electrical Level and 0-3 Documentation levels
Power/Ground Plane	1 Electrical level and 0-3 Documentation levels

NOTE: Silkscreen and Assembly Drawing already have their physical or Electrical level pre-selected and hence only Documentation levels need be selectable.

A Power/Ground plane need not have an electrical level selected. If no electrical level is selected, then the pad sizes are determined by the pad stack values for the first documentation level selected. If an electrical

level is selected then the pad sizes are determined by the pad stack values for that level. In any case, the size of the thermal relief pads at the "connected" pins are the same size as the pad stack value but with a thermal relief shape substituted in place of the shape in the pad stack. This Thermal Relief pad serves to connect the pin to the plane when reverse contact printed.

After picking a PLOT TYPE, the Default levels for that Plot Type are shown already selected. Additional levels may be selected or de-selected, as necessary.

Select the level(s) desired on the PLOT TYPE and proceed to the Decision area and select "Action" (PROCEED, GO BACK or EXIT). PROCEED will result in the next CRT screen of SELECT ITEMS as shown in Figure 11.5.

Select Items						
ITEMS		ITEMS	L1	L2	L16	L20
Board		Pads				
Connections		Vias				
Parts-Top		Tracks				
Parts-Botm		Copper Areas				
REF Designators		Line-Items				
Part Types		Text				
		Part Outlines				
PROCEED	GO-BACK		EXIT			

Figure 11.5 - SELECT ITEMS

The SELECT ITEM CRT screen indicates the items (Highlighted) that have been predefined for a specific plot type, and also permit the designer to change the item for the plot type being processed. When a specific Plot type is selected, a preset number of items will be associated

plotted with the design origin 1 inch up and to the right of the plotter origin. Mirroring and Rotating functions from the design origin.

NOTE: Offsets must be calculated taking previous Transformation Options into account, i.e., Scaling, Rotation, and Mirroring are applied before offsetting.

- ON-LINE PLOT (Y/N):Y

  This allows you to override the Default On-Line or Off-Line setting for the output device chosen during installation of the software. Selecting "Y" will cause the output to go directly to the output device (printer, pen plotter or photoplotter). Selecting "N" will cause the output to go to a disk file. This disk file may be sent to a plotter at a later time using the standard DOS "Copy" command.

- PLOT JOBNAME (Y/N):Y

  Selecting "Y" will cause the JobName, Time and Date to be plotted on the drawing. Selecting "N" will suppress the output of the JobName, Time and Date.

- SUPPRESS REF-DESIGNATORS STARTING WITH: ____

  Is displayed ONLY if the Ref. Designation was selected during "Select Items". A response with the first letter of the Ref Designator, permits the suppressing of the Ref Designators starting with that letter.

Example: You have Mounting Holes and Moires on the board using a Ref Designation starting with M, i.e., MH1, MH2, MOIRE1 etc. A response of M would suppress those Ref Designations on the plot, but all other Ref Designators would be plotted. Up to 7 different starting letters may be specified. Enter the prefix(es), no spaces or delimeters are required.

- OVERSIZE AMOUNT FOR PADS: 10

  This is displayed only if Solder Mask was selected. All pads are plotted with their sizes increased by the amount specified. Enter the amount required if different from the default value. Both positive and negative integers are allowed.

- SUPPRESS PARTS WITH REFS STARTING WITH:__

  This is displayed only for the SMD Paste Mask plot. Respond with the first letters of the reference designators of the parts to be de-selected from the SMD Paste Mask.

- PLOTTER PEN WIDTH: 16

This is displayed only if Pen Plotter was selected on the Output Device Screen. Change this value to reflect the actual pen size used for the plot. If this is not specified correctly then solid filled items such as pads, tracks and copper may not be drawn correctly.

If either Power or Ground Planes have been selected, following the Transformation Option screen of Figure 11.6 will be the ENTER POWER/ GROUND SIGNAL NAMES screen of Figure 11.8

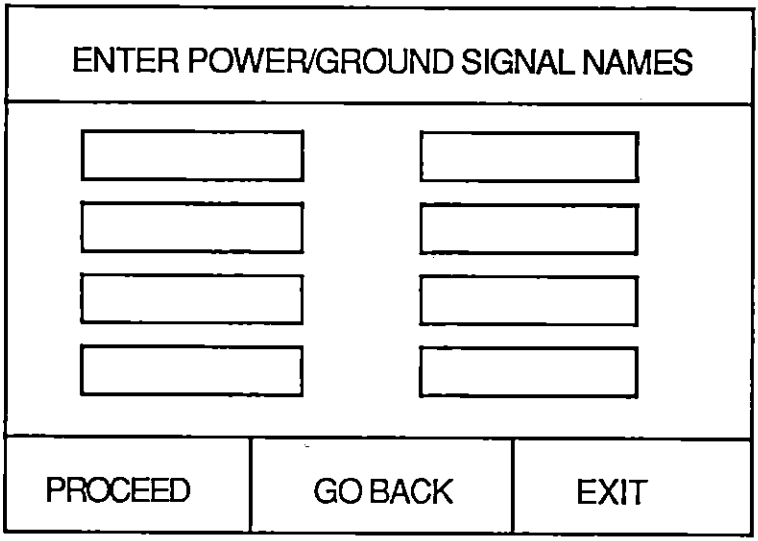


Figure 11.8 - ENTER POWER/GROUND SIGNAL NAMES

Signal names selected (i.e., VCC, PWR, +5V, etc.) will flash a Thermal Relief pad on the selected level for the particular Net. This Thermal Relief pad will connect the pin to the plane when reverse contact printed. Up to 8 different Signal names may be selected. In order for this to work correctly, the appropriate Power and Ground connections must be in the database. The connections should be left unrouted but they have to be present in order for the software to recognize the pads that must be connected to the plane.

Thermal Relief pad sizes are determined by the pad size defined for the electrical level, or the first level specified in the Power/Ground plane level selection.

After selecting the Transformation Options or after selecting Power and Ground Signal names for a Power/Ground Plane plot, the menu shown in Figure 11.9 will appear.

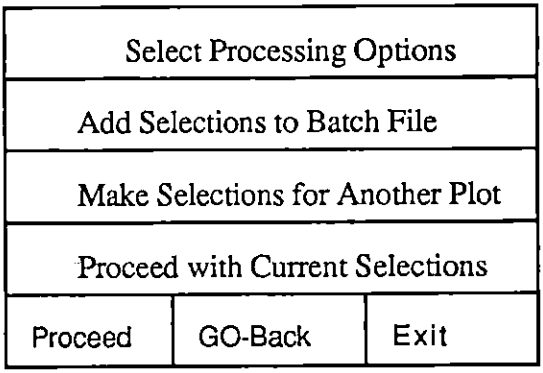


Figure 11.9 - PROCESSING OPTIONS

This is the last menu for the DIRECT processing option. The "Add Selections to Batch File" box is already highlighted meaning that all the selections made in the previous menus will be stored in a "Batch File" for your job. This means that this plot can be recalled anytime later using the BATCH option. If this box is de-selected, the plot selections will not be stored in the Batch File.

The "Make Selections for Another Plot" causes your current selections to be saved in a temporary file and return to the "Select Output Device Menu". Any number of plots may be "stacked" up before actually processing the plots.

When the "Proceed with Current Selections" box is picked, this plot and all previously stacked up plots will be processed.

The system will go busy for anywhere from a few seconds to several minutes, depending on the complexity of the selected plots. Post processing can be cancelled at anytime by pressing Function Key 10.

NOTE: It may take a few seconds for the system to respond to the F10 Key. Also note that the right mouse button does not work as F10; the actual F10 Key must be pressed to cancel a plot.

When processing has completed or has been cancelled, the system will return to the menu in Figure 11.1

If a plot is being generated for the photoplotter, a report titled "Photoplotter Apertures Report" will be printed. This lists the apertures used by the plot. If the plot requires aperture sizes that are not available in the "Aperture Table" (see section on Default Options), a list of the apertures used is printed followed by a list of the unavailable aperture sizes required. The system will then request that the user enter an aperture number to be used for each of the unavailable sizes as follows:

Enter Aperture No. for Width: nn, Height: nn, Shape: xxxx

These new aperture assignments are made temporarily for this plot only, and do not affect the "Aperture Table". To make these temporary assignments you will need a print-out of your "Aperture Table", which you would have obtained last time you updated the Aperture Table in the Defaults section.

Suggested method of picking aperture assignments:

(a) Choose an unassigned aperture if available.

(b) If no unassigned apertures remaining, then choose one that is not used for this plot.

(c) If no more unused apertures, choose a used one that has the same shape and is closest in size.

(d) If there are no suitable apertures available, the plot must be cancelled (use F10 Key) and the Pad Stacks or Line Widths in the design must be modified to use fewer different sizes.

11.5 Batching CAM Outputs

The BATCH Routine is strictly a file of previous CAM Selections (not actual plots/files), thus allowing the user the ability to redo previously generated plots. This is particularly useful after a revision has been made to the board since all the previous parameter selections for the post processing of the original design have been saved in the Batch File for the job. All the plots for a job can be recreated without having to go through all the menus in the DIRECT processing option even though the database for the board has been changed during the incorporation of the revision. The Batch File is named BATCH.DAT in the subdirectory selected for the board. The Batch File is automatically created the first time a plot is added to the Batch File in the Processing Options menu (see Figure 11.9). When the BATCH option is selected, the menu shown in Figure 11.10 is displayed.

Select Batch Option								
LIST	1	2	3	4	5	6	7	8
	9	10	11	12	13	14	15	16
DELETE	17	18	19	20				
RUN								
PROCEED	G0-BACK				EXIT			

Figure 11.10 - BATCH OPTIONS MENU

There are three main functions that can be picked with the mouse:

LIST
This prints out a list of all the plot selections stored in the Batch File. The plots are sorted by Output Device, Plot Type and Level numbers.

The printout includes a numerical index for each plot which is required in the other two functions DELETE and RUN.

DELETE
This option allows deletion of unwanted plot selections from the Batch File. When this option is picked, a grid of all the plot indices appears in the menu. Figure 11.10 shows an example with 20 plots. All the unwanted plots should be picked by selecting the plot indices from the printout obtained from the LIST option. After picking all the indices, select the PROCEED box. A new listing of plots reflecting the deletions is automatically generated.

RUN
This option allows previously saved plot selections to be reprocessed. When this option is picked, all the plot indices are displayed as with the DELETE option. Any number of plot selections may be picked after which, selection of PROCEED will cause all the plot selections selected to be processed sequentially.

11.6 Changing CAM Defaults

The DEFAULTS option provides the ability to change certain default parameters and tables for post processing of a job. The defaults are stored both in the master defaults file: DEFAULTS.PAD and in the Batch File for the job. The defaults may change from job to job. When the "Defaults" menu is selected the menu below will be displayed:

1 Misc	**2 Pho-Plot**	**3 N/C Drill**	**4**	**5**
6	**7**	**8**	**9**	**0 Exit**

11.6.1 Defaults - Misc

When the "MISC" option is selected, the menu as shown in Figure 11.11 is displayed.

The "MISC" option allows modification of the "miscellaneous defaults including the drill marker symbols for the Drill Drawing plots.

The "Pho-Plot" option allows modification of the Photoplotter Aperture Table.

Enter Drill Feed & Speed Table Modifications:

Size	Range	Feed Rate	Drill Speed
20	29	200	500
30	39	400	500
40	49	500	500
50	59	500	450
60	69	400	400
70	79	400	350
80	89	380	300
90	99	350	250
100	119	300	250
120	139	250	250
140	159	200	250
160	199	150	250

PROCEED	GO BACK	EXIT

Figure 11.13 Drill Feed and Speed Table

This menu allows the specification of up to twelve different combinations of speed and feed parameters each allocated to a range of drill sizes. The values shown above or the values as they first appear when you first enter this menu do not necessarily represent meaningful values for your drill machine. You must enter the correct values for your machine; otherwise, you risk damage to the board or you may break the drills in the machine. Talk to your drill house and obtain the speed and feed parameters for different drills. You can suppress the output of feed and speed commands in your N/C Drill output by modifying the DRILL.DAT file explained in Section 11.11. If you decide to use this feature, then simply enter the correct feed and speed parameters. The drill size ranges are also purely arbitrary, hence you may alter them also.

11.7 Verify

The VERIFY option provides the ability to check your Gerber output files by reading them back into the system and plotting them on either your pen plotter or dot matrix printer.

When the VERIFY option is selected, it will prompt you for the file name of the Gerber file you wish to check with the message:

SPECIFY GERBER FILE NAME:

The response should be just the file name portion of the Gerber file, i.e., no extension or directory path name. For example, ART01 for a level one Artwork file.

After entering the file name, the Output Device Menu as shown in Figure 11.2 will appear. Either the Pen plotter, Matrix printer or Laser printer device may be picked. (It makes no sense to select Photoplotter again!)

After selecting the Output Device, the Transformation Options Screen as shown in Figure 11.7 will appear.

NOTE: None of the other screens from the DIRECT option will appear since we are not extracting any data from the database. In fact, it is not necessary to have any job database in memory at the time the VERIFY option is being used.

The Transformation options for the plot should be entered in accordance to the instructions in the DIRECT option. Having entered the Transformation options and selected PROCEED, the required plot will be generated on the selected output device. When the plot is completed, the system will return to the menu in Figure 11.1.

11.8 CAM Output Files

If the "ON-LINE PLOT (Y/N):" option in the Transformation Options Menu (Figure 11.7) was set to "N" then the output that would normally be sent directly to the Printer or Plotter would be stored as a file on disk. Photoplotter output is normally stored as a disk file, but the Matrix and

Laser Printer, and Pen Plotter output can be stored in a file also if desired.

The disk filename is chosen automatically for you, but you are able to specify in which sub-directory these files are to be stored. This is done with the very first prompt that is made when entering the CAM menu, i.e.:

SPECIFY CAM OUTPUT SUBDIRECTORY

The sub-directory that you specify is created in the "CAM" sub-directory which is located in the "PADS" directory. The filenames chosen by the CAM menu for your output files reflect the contents of the file.

The first two or three letters represent the Plot Type selected as follows:

ART	Artwork Plot
SST	Silkscreen Top Side
SSB	Silkscreen Bottom Side
ADT	Assembly Drawing Top Side
ADB	Assembly Drawing Bottom Side
DD	Drill Drawing
SM	Solder Mask
PGP	Power/Ground Plane
GEN	General Plot
SMD	SMD Paste Mask

The next two or four digits represent the Level or first two levels selected for the plot with values ranging from 01 to 30 for each level.

If the character "A" appears after the Level digits, this indicates the file contains data for an "Artwork Quality" pen plot.

The filename extension indicates the Output Device selected as follows:

.PHO	Photoplotter
.MPR	Matrix Printer
.PXY	Pen Plotter
.LPR	Laser Printer

where "XY" are digits indicating pen number and sheet number.

Pen Plotter outputs may have to be stored in several files for each plot, hence the "XY" digits in the extension. If the plot occupies more than one sheet of paper, then a separate file has to be created for each sheet to allow the user the opportunity to change sheets when plotting directly from a file. Also, pen plotters that do not have an automatic pen change, require a separate file for each pen to allow the user the opportunity to change pens. The "Y" digit indicates the sheet number and the "X" digit indicates pen number. If X=0, then the plotter has automatic pen change.

Example output filenames are:

ART01.PHO	for Artwork, Level 1 for Photoplotter
DD0224.MPR	for Drill Drawing, Levels 2 & 24 for Matrix printer
GEN03A.P01	for General Plot, Level 3, Artwork Quality, Pen Plot Sheet #1

NOTE: Output files for Printers and Pen Plotters can be extremely large, up to one megabyte in size, for large boards, hence, it is desirable to generate plots on-line directly from the CAM menu whenever possible to avoid filling up your disk with these large output files. Fortunately, Photoplotter files are more reasonable in size and unless you have the luxury of a direct on-line link to your photoplotter, you will need to generate the output files. Normally you would copy your photoplotter files to diskettes and send your diskettes to a Service Bureau that will accept Gerber photoplotter data on MS-DOS diskettes.

11.9 Wet Ink Quality Plots

HP or HI wet ink plotters can be used for creating either check plots or high quality plots with the tracks and pads filled such that the plot can be photo reduced to 1:1 final artwork.

There are three elements to making good plots that are to be used for final artwork:

1) the correct pen
2) the correct paper
3) the correct ink

Pens must be the correct width and long lasting. Both Staedler and KOH-I-NOOR pens are excellent and are available at most drafting supply houses. Consider the following:

2:1 plots Staedler 757P66CS
KOH-I-NOOR 63VIB 2/.60

1:1 plots Staedler 757PL3CS
KOH-I-NOOR 63VTB-001.30

INK

The printing characteristics for INK are as follows:

(1) to flow freely during the plotting strokes
(2) not drying up when not plotting

The following INK vendors are suggested:

1. Graphics Resources Intense Black Plotter Ink
2. KOH-I-NOOR 3084-F Rapid Draw Ink

PLOTTING PAPER

There are many types of plotting paper available. Both double or single matte polyester are satisfactory. Use paper of at least .003" thickness to avoid stretch. One suggested paper is Dura-lene Polyester Matte Plotting Paper.

SUPPLIES

Suggestion: Keep your pens clean! To do this you will need an array of cleaning materials including an ultrasonic cleaner, if you budget will permit. Discuss with your drafting supply dealer and he will make recommendations.

11.10 The N/C Drill Output

The N/C Drill command outputs an ASCII file to drive a range of drilling machines. The N/C Drill Menu is selected by selecting the CAM Menu, then entering a directory name in response to the "CAM

Output Subdirectory" prompt, and then selecting the "Direct" option from the Menu in Figure 11.1. So far, this is identical to the rest of the CAM software. After selecting F1 for "Direct", the "CAM Output Devices" Menu in Figure 11.2 will appear which provides the option to select N/C Drill Output. This command allows you to generate either or both of two output files:

(1) A file containing a listing of the drill hole co-ordinates and their sizes on the board which can be output to a printer.

(2) A file containing commands to drive a numerically controlled N/C drilling machine.

The N/C command file will drive the Excellon series of drill machines or other compatible models. However, the user has the ability to vary the format of the output file by editing the contents of the Drill Format File DRILL.DAT in the "\PADS" Directory. (See Section 11.11).

BURIED VIAS

The N/C Drill command will output all the drill holes resulting from normal (i.e., not surface mounted) component pads, through vias (Standard or Micro size), and buried vias. The treatment of component pads and through vias is obvious: either you select them or you don't! However the treatment of buried or blind vias requires some explanation. Buried via drill holes are selected by level or they can be totally de-selected. Two levels must be selected which will cause only the buried vias starting and ending on the levels selected to be output. For example, if the user selects levels 2 and 4 he will only get the holes for vias starting on level 2 and ending on level 4. Holes starting on level 2 and ending on level 3 or 5 will not be output, nor will the holes starting on levels 1 or 3 and ending on level 4 be output. In the instance of a 6 layer board with a Buried via pair on Layer 1/2 and 5/6, make two separate passes with the first pass for Level 1/2 and the second pass for Level 5/6

N/C DRILL MENUS

After selecting the N/C Drill option on the "CAM Output Devices" Menu the menu in Figure 11.14 shown below, will appear. NOTE: The level entry appears only if Buried Vias are selected.

This menu allows you to select the types of drill holes to be included in your output. The menu allows selection of any combination of the following types of drill holes:

a) Drilled component pads
b) Through vias
c) Buried vias

The menu is displayed with pads and through vias already selected for convenience. Selection of buried vias causes the level numbers to be displayed on the right hand side of the menu. As described earlier, only buried vias are selected by level. Only the routing levels are displayed for your board. You must pick two routing levels for buried vias. After picking the drill hole types, you can proceed to the next menu by picking the "PROCEED" box, after which, the menu shown in Figure 11.14 will appear.

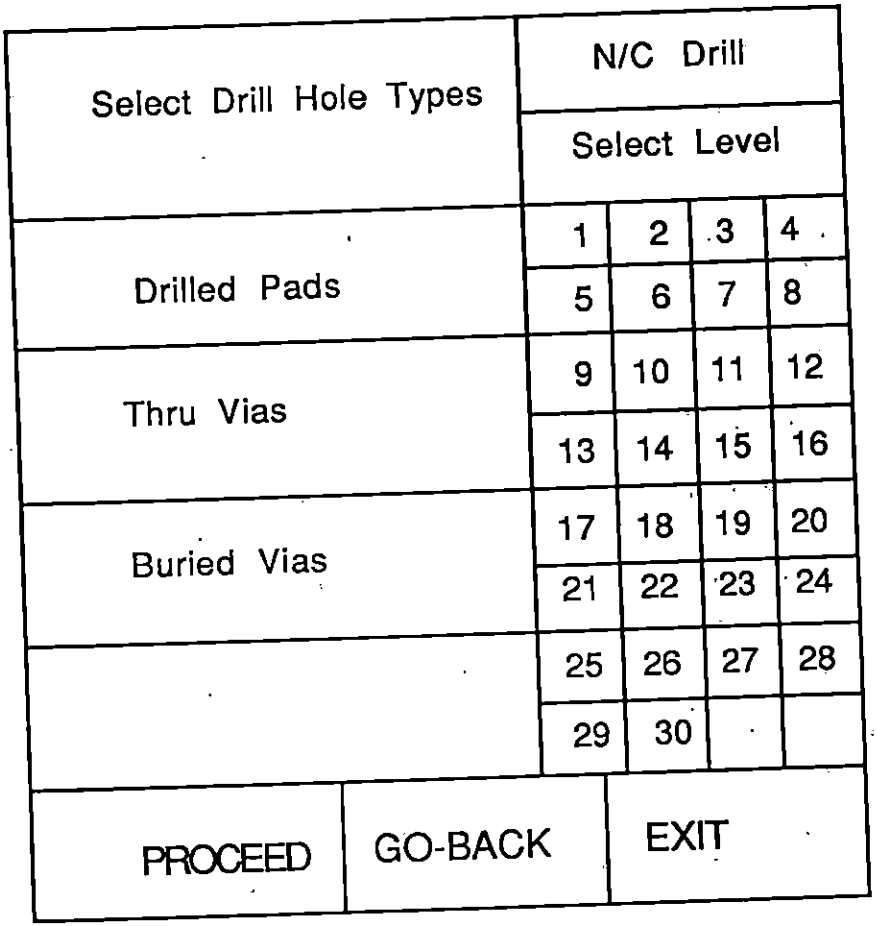


Figure 11.14 Drill Hole Type Selection Menu

The following describes each of the lines of data in the above example format of a \10.

LINE 1:

The first line (1) of Part Type data for the full format contains 9 items as follows:

PART TYPE NAME PCB DECAL NAME LOGIC FAMILY LINES GATES SIGNAL PINS PIN NAMES CONNECTOR IDENTIFICATION and NUMBER OF CONNECTOR PINS

Where PART TYPE NAME is the name of the Part Type, i.e. 68000, R1/4W, etc.,-Part Type Name is also the entry in the Part Type location on the Part Decal, thus:

and is associated with the Reference Designation when creating an Input Parts List to create the database for a board. A Part Type name can have up to 16 alphanumeric characters - it may NOT have special characters, however, the following are illegal: * ( ) { } [ ] :

NOTE: Non TTL Digital components are normally named by the vendors Specification Data Sheet, i.e., 8088, 68000, etc. TTL parts of the 54 and 74 families are normally named by a Back Slash followed by the Vendor's Part Number that uniquely identifies the part, i.e., \10, \128, etc. This is referred to as a Suffix. Parts using a Suffix Part Type Name always require a PREFIX LINE in the Part Type Information.

Integrated circuits, especially TTL IC's may, in the interest of maximizing the contents of the Library without increasing the data handling requirement, use a concept called Suffix and Prefix. The use of a Suffix and Prefix for a 54, or 74 series part is signaled by two items:

1) The first character of the Part Type Suffix will be a backslash thus: \.

2) The numbers following the backslash are those used to uniquely identify a device: 00 for a 7400, 192 for a 54LS192 etc., etc.

Discrete Parts, because of the multi-character nomenclature and potentially large number of items in the Part Library use a "Family" and "Tag" need a "coding" convention. See end of this chapter for suggestions (Section 12.9 Discrete Part Type Naming Convention).

The Part Type Name for Discrete Parts can be R1/4W or R2W, etc. (See Paragraph 12.9).

DECAL NAME is the name of the Part Decal used to create the footprint of a part, as displayed at the CRT or plotted, i.e., DIP14. Up to 16 characters can be used. These first two entries (Part Type name and its associated Part Decal) establish the necessary link or pointer between the two items. Each part type MUST have an associated Part Decal that has been named in the Part Type header AND the Part Decal MUST be in the Part Decal Library. (Either the User Part Type area or Standard Part Type area).

Up to four different (alternate) PCB Decals may be associated with a Part Type. When more than one Part Decal is used, each Part Decal is separated by a colon, thus:

PART NAME WWW:XXX:YYY:ZZZ TTL etc.

Different Part Decal names are used in the ALTERNATE command of the PLACE menu so as to allow the designer to change the Part Decal during placement of components. When a part is brought into the database, the first Part Decal in the Part Type is brought up during the component placement. The selection of ALTERNATE will substitute the next Part Decal for the first, and each succession selection of ALTERNATE will select the next Part Decal associated with the Part Type. (See Place, Section 6.2.5).

The different or alternate Part Decals associated with the Part Type are created in the Create Part Decal command and stored with different Part Decal Names. They can be all located in the User Part Decal area, the

Standard Part Decal area, or they can be located in both the User area and the Standard area.

Each Alternate Part Decal used in a Part Type MUST have the same number of terminal pins - if not, an error message will result when the alternate part decal is requested.

LOGIC FAMILY is the logic type as follows:

RES R
TTL U
CMO U
ECL U
ANA U
CAP C
DIO D
ZEN Z
TRX Q
XFR T
CHO L
CON J
RLY RL
SCR TH
TRC TH
SWI SW

NOTE: In addition to indicating the Logic Type, the LOGIC FAMILY entry performs a second function as described below.

The AutoRouter will not place vias under DIPS or IC Components if you indicate that you do not wish vias under IC's, as you are setting up the AutoRouter. When you have commanded "No Vias under IC's", any component that has the Logic Type of TTL, CMOS or ECL will not have vias under its outline. Components that have the Logic Type ANA or UND will allow vias under their outlines.

LINES* (optional) Following LOGIC FAMILY, there are six data entries as follows:

NUMBER of lines of user text, including data to follow (an unlimited amount of lines allowed).

GATES** (optional) Used as the basis for Gate Swapping. Defines the number of gates in the Part Type (up to 99 allowable gates per part). In determining the entry for number of "gates" in a Part Type, count the number of swappable gates. A 74LS00, for example, would have 4 gates; A 7410 would have 3 gates. If there are no swappable gates, enter a Zero (0); a Resistor would have 1.

SIG NAMES*** (optional) Enter the number of pins on the component that will have their Signal Names built into the Part Type Library. The Signal Names are used to originate connection nets using an automatic method available in PADS. Example: *SIG* +5V in the connection list would extract all pins using the signal name +5V on all parts on the board, and create a Net named +5V.

PIN NAMES+1 (optional) Used to specify the number of Alpha Numeric pin names to terminal pads on a component, such as would occur with the terminal of a pin grid array, i.e., A1 B1 etc.

CONNECTOR FLAG+2 (optional) Enter a 1 IF the part is a connector, otherwise enter a 0.

CONNECTOR PINS+3 (optional) If a connector, then enter the Number of Pins to be associated with the connector, i.e., 26, 100, etc.

PREFIX LINE (Line 2):

Part Type Prefixes are used for TTL parts of the 54 & 74 series, so as to include all variations of these parts, without an individual entry in the Library for each variation.

The first character entry is the PREFIX Line is @. The next entry may be X4, where X is a shorthand parameter for both 5 & 7, i.e., 54 & 74. The X4 is used only when the part has both 54---and 74--- variations.

Next, is listed the variations of the device, i.e., ALS, L, H, 5, etc.. If both the 54 & 74 series has the variation of ALS, enter as X4ALS (Not 54ALS and 74ALS).

If only the 54-- series has a variation L for example, enter 54L, (but not X4L, or 74L).

In addition to the PREFIX LINES, there are additional data lines with HEADER IDENTS used as pointers for creating reports, parts lists, priced bills of material. The headers are as follows:

PART DESC	Header for a line of data describing the Part, i.e., 4 bit binary counter, etc.
$XXX.XX	Unit cost
PART NUMBER	16 characters to be used for either vendor catalog number or internal company identification.
MFG #1	Vendor name
MFG #2	Alternate vendor

The above data lines, 3-7, are individually and collectively non-mandatory and used only at the user's option.

NOTE: Although you may decide not to build your Part Type Library using the additional data lines and even "gating" information, we strongly recommend you include Standard Power and Ground Signal Names in your Part Type Library. This data can be used to automatically create the Power and Ground Connection List of your board.

In the format example at the start of this discussion, the first line of data (1) defines Part Type \10, and uses Part Decal DIP14, is TTL Logic, will have 6 lines of additional text , has a total of 3 gates, 2 pins with Signal Names and no Pin Names.

Following the first line of data in the Format example, are the six lines of additional data on the Part (Lines 2-7) as indicated by the LINES entry of the first data line. In the example, the six lines provide ALIAS information, cost data, vendors, company part name, etc.

Gate Lines (8), (10), and (12).

The first line of each Gate entry (8) (10) & (12) consists of the following:

G:DECAL GSWPTY PINS

where G is the gate line Header, with G used for a swappable gate, and where DECAL is the name of the CAE Decal used for the gate.

GSWPTY is an arbitrary non-zero number used in each gate of a Part Type to indicate that this gate can be swapped with other gates using the same Gate Swap Type Number, either within the same component or other parts of the same Part Type on the board. A "0" entry will "freeze" the gate during Gate Swapping so that the gate will NOT be swapped.

NOTE: Swapping will NOT occur on swappable gates/pins of a component that have been glued down.

Pins is the number of pins in the gate.

The first gate line of the format example (8) reads G:NAND3 1 4, and is interpreted as a gate using the NAND3 Decal that can be swapped with any gate, using a GSWPTY Number of 1, in all unglued \10 Part Types on the board, and the gate contains 4 pins.

PIN DATA LINE

The second line of gate data (9) (11) & (13) for a gate consists of the following information on each pin in the gate.

PIN NO. PADSWAP PIN TYPE

Where PIN NO. is the number of the pin on the Part Decal.

PADSWAP is an arbitrary non-zero number ("1"), indicating the pin can be swapped with another pin of the same pin swap type within the same gate, but not between gates of the same Part or other unglued \10 parts on the board. A "0" indicates the pin cannot be swapped with other pins in the gate.

The second line of data in the format example (8) on the first gate reads:

1.1.L 2.1.L 13.1.L 12.0.S

and is interpreted as follows:

Pins 1, 2 and 13 are defined as load and swappable within the gate of the part that they are located in or on. Pin 12 is the 4th pin (source) on the same gate and is not swappable with any other pin on the gate of which it is a part.

NOTE: The use of a space to separate the data associated with each pin, and the use of a period (.) to separate the data associated with a pin is required, thus the second line of the second Gate (line 10) reads:

3.1.L 4.1.L 5.1.L 6.0.S

NOTE: In addition to being logically swappable, the pins within a gate must be arranged physically symmetrical. That is, if the first gate has input pins arranged first, followed by the output pins, then all other gate pins must be entered in the same symmetrical fashion, i.e., inputs first followed by outputs.

PINTYPE is the electrical type of Pin. There are ten (10) allowable entries, as follows:

S = Source
B = Bi-directional
C = Open Collector
O = OR-Tieable
T = Tri-state
L = Load Pin
Z = Terminating Resistor
P = Power
G = Ground
U = Undefined

The pin type is separated from the other pin data by a period.

LINES 14 & 15:

The data line (14 & 15) defining pins with specific Signal Names in the Part Type is as follows for each named pin:

SIG PIN PIN NO. TRKWIDTH SIG NAME

where SIG PIN is Line Header

PIN NO. is Part Pin Number. Max character length is four (4).

TRKWIDTH is the width (in thousandths of an inch) of the track connected to the pin (normally .050" for power and ground). Minimum value allowed is 1; maximum value allowed is 250.

SIG NAME is the Name of the Signal connection made to the pin. Max character length is twelve (12)

The first Signal Pin Line (14) of the format example reads: SIGPIN 7 50 GND to indicate "pin 7 is connected with a .050" track to a connection Net named GND." Note the use of a space to separate data (mandatory).

The second Signal Pin Line (15) reads:
SIGPIN 14 50 +5V to indicate "pin 14 is connected to a .050" track that is part of a connection Net named +5V."

LINE 16:

The PIN NAME entry (Line 16) is the last entry in the Part Type data. It is used to provide an alpha numerical pin name to a terminal as is required in a Pin Grid Array component, rather than numerics only. The entry is A1, B2, C12, etc as per the Pin Grid Array Vendors Specification Drawing, where the first pin name is the name corresponding to terminal One on the Part Decal. A one-to-one correspondence between the pin names in the part type and to pin numbers in the Part Decal is required. Maximum character length is 4.

NOTE: A space - not a comma - separates each PIN NAME

12.6 PART TYPE (CREATE, PART TYPE Format)

In the previous section, the creation of the Part Type information using PLIB was described. An alternate method using the PART TYPE subroutine of the CREATE Routine, within the PADS-2000 application program, may also be used.

When PART TYPE (F9) of CREATE is selected, the following menu appears:

```
      Gate Information for Part \01
Gate A:  Sw  Pins  PADS-LOGIC Gate Decal & Alternates (not req'd for PCB)
          1    3        NAND20C  NANDDMOC
Pin #  T  SW Pin-Name   Pin #  T  SW Pin-Name   Pin #  T  SW Pin-Name
  3    L   1             2    L    1            1     CO   OC
Gate B:  Sw  Pins  PADS-LOGIC Gate Decal & Alternates (not req'd for PCB)
          1    3        NAND20C  NANDDMOC
Pin #  T  SW Pin-Name   Pin #  T  SW Pin-Name   Pin #  T  SW Pin-Name
  5    L   1             6    L    1            4     CO   OC
Gate C:  Sw  Pins  PADS-LOGIC Gate Decal & Alternates (not req'd for PCB)
          1    3        NAND20C  NANDDMOC
Pin #  T  SW Pin-Name   Pin #  T  SW Pin-Name   Pin #  T  SW Pin-Name
  8    L   1             9    L    1            10    CO   OC
Gate D:  Sw  Pins  PADS-LOGIC Gate Decal & Alternates (not req'd for PCB)
          1    3        NAND20C  NANDDMOC
Pin #  T  SW Pin-Name   Pin #  T  SW Pin-Name   Pin #  T  SW Pin-Name
  11   L   1             12   L    1            13    CO   OC
```

Figure 12.5 - GATE INFO FOR A 7401

The data entered in the GATE INFO is the same as contained in the GATE INFO in PLIB.

12.6.3 NEW CONN

When NEW CONN (F3) is selected and the New Connector named, the PART INFO (F1) and GATE INFO (F2) menu is presented. (The GATE INFO (F2) is inactive for Connectors.)

The CRT presentation for NEW CONNECTIONS, PART INFO (F1) is as follows:

```
Connector Part Type:                X
Number of Connector Pins:           0
Logic Family:                       CON
PCB Decal & Alternates:
   XX
Number of Part Attributes:          0
Non-Numeric Pin Nos. (Y/N):         N     Lines Required: 0
```

Figure 12.7 - CRT PRESENTATION FOR NEW CONN, PART INFO (F1)

A completed PART INFO for a Conn/26P/Ed is as shown below:

```
Connector Part Type:                CON\26P\ED
Number of Connector Pins:           26
Logic Family:                       CONN
PCB Decal & Alternates:
   CON\26P\ED
Number of Part Attributes:          5
   PART DESC: 26 PIN EDGE CONNECTOR
   $:
   PART NUMBER:
   MFG. #1:
   MFG. #2:
Non-Numeric Pin Nos. (Y/N):         N     Lines Required: 0
```

Figure 12.8 - PART INFO FOR A CONN/26P/ED

12.7 Initialize

This command can be used to clear out or erase the contents of the entire Standard area or User area including Lines, Decals, and Part Types.

To use, select INITIALIZE, and confirm you desire to initialize PLIB. All items in the four sublibraries of the area selected will be erased, leaving this area of PLIB empty.

CAUTION: This is an "Extreme" command. Be sure you wish to clear your Library before using!!

12.8 System Parts Library

A comprehensive selection of Line Items, Part and CAE Decals and Part Types are included with the PADS-2000 Program. These are included in the Standard Part area. A Print Out of the Directory of these items can be obtained by selecting the Standard area, using the "List Directory" command (L) and responding with * for ALL, as outlined in 12.5.1.

12.9 Maintenance of Systems Parts Library

12.9.1 Creating or Modifying Part Decal/Types:

The system parts library which includes lines, part decals, schematic symbols, and part types with your system should be kept on the hard disk. At various times you will need to add a new part decal or part type to the library. Use the CREATE DECAL command for Part Decals. The new part decal will be automatically placed in the User area of the library. When you wish to MODIFY a Part Decal stored in the Standard area, the SAVE function will save the modified decal in the Standard area unless the S:XXX name is replaced with U:XXX. In the later instance, the part decal will be stored in the User area with the name XXX.

Use either PLIB or CREATE, PART TYPE to create/modify Part Types. Part Types are stored in the User or Standard area in the same fashion as are Decals.

It is recommended that NEW Part Types created by the user, be created and stored in the User area. Part Types originally supplied by CAD Software, Inc., in the Standard area, and modified by the user, should be modified and stored in the Standard area of PLIB.

12.9.2 Making Backups of the Library

We wish to emphasize that you should maintain a current backup copy of all Part Decal and Part Types. PERIODICALLY, (AT LEAST ONCE PER MONTH), YOU SHOULD FIRST CREATE A BACKUP FILE ON FLOPPY DISK IN THE EVENT A PROBLEM ARISES DURING UPDATING.

Individual library items cannot be transferred to and from disk storage as individual file transfers. The entire System Parts Library file must be copied for storage as a whole rather than on a record-by-record basis. Similarly the entire file is read from floppy disk to hard disk for usage. In the process any records in the hard disk Library are over-written by the file on floppy disk. Be sure you take this into consideration in planning your Library updating effort.

Saving the Part Types Library

To copy the existing Part Type records FROM the subdirectory LIB TO a floppy disk proceed as follows:

Place a floppy in the destination drive (in this example drive A:), enter the subdirectory LIB by entering the DOS commands:

```
CD\PADS\LIB <CR>
COPY UPART.*  A:<CR>
COPY SPART.*  A:<CR>
```

This will copy four files from your hard disk to floppy drive A: UPART.DAT, UPART.IDX, SPART.DAT and SPART.IDX.

Saving the Part Decal Library

To copy the existing Part Decal records FROM the subdirectory LIB TO a floppy disk proceed as follows:

Place a floppy in the destination drive (in this example drive A:), enter the subdirectory LIB by entering the command:

```
CD\PADS\LIB <CR>
COPY UDECAL.* A:<CR>
COPY SDECAL.* A:<CR>
```

This will copy four files (part decal data and index) from your hard disk to floppy drive A: UDECAL.DAT, UDECAL.IDX, SDECAL.DAT and SDECAL.IDX.

Saving the 2-D Lines Library

To copy the existing Line Item records FROM the subdirectory LIB TO a floppy disk proceed as follows:

Place a floppy in the destination drive (in this example drive A:), enter the subdirectory LIB by entering the command:

```
CD\PADS\LIB <CR>
COPY ULINE.* A:<CR>
COPY SLINE.* A:<CR>
```

This will copy two four (2-D Line data and index) from your hard disk to floppy drive: ULINE.DAT, ULINE.IDX, SLINE.DAT and SLINE.IDX.

WARNING: The Lines Library must be backed up when backing up the Decal Library, even if no modifications have been made to the Library. This is because the Decal Library has "pointers" to the Lines Library.

12.10 Discrete Part Naming Convention

The naming of the Part Type for discrete components is a special case and is slightly different from the Part Name naming of Digital components. The reason for this somewhat different technique is the great number of discrete parts (perhaps in excess of 20,000), if each discrete part were uniquely identified in the PLIB Library, down to its VALUE and TOLERANCE.

The technique used for discretes is called the "Family and Tag" approach. In this approach, each category of discretes (Resistors, Caps) is grouped into families in accordance with a common Decal and Tags (Value and Tolerance) are added to uniquely identify the part. In the instance of Resistors, the wattage is used to establish the family: R1/4W is the part type name for quarter watt resistors, R1W is for one

watt resistors. Families of Caps are CK XX, CL XX, etc. The Part Type name for the component is the family value and the Part Decal uses the SAME name. Consequently, the first line of the Part Type data would read:

```
R1/4W R1/4W RES 000000
```

Where the first R1/4W is the Part Type name and the second R1/4W is the Part Decal name.

Consequently, to call up on the board the part type for a part, use the Part Type Family name i.e., R1 1/4W and a part type and decal for a quarter type resistor will be brought into the board database.

Most firms wish to uniquely identify the part including the data on a parts list for adding the part. To do this, we use a concept called "Tags" together with the part family name to result in the complete part name. "Tags" are applied at the time the part is brought into the database using FutureNet, a Parts List, ASCII IN, Design-On-The-Fly and ECO. The utilization is as follows: Enter the part type family name followed by a comma, followed by the "Tags". These tags contain the additional data needed to uniquely identify the part and should not contain a comma. The following is a typical entry into the Parts List to call up and uniquely identify a part.

```
R1   R1/4W,100K,5%
```

This entry requests a 100K OHM, ± 5% (Value and Tolerance) resistor. The Tags are attached to the part when the part type/decal is brought into the board database, and remain with the part while in the database. The complete part type identification is displayed at the CRT as "Part Type" and is printed on the parts list.

CAUTION! The sum total of the Alpha Numeric and Special Characters that can be used for BOTH the Part Type Family and Tags cannot exceed sixteen.

NOTE: All the Discrete Parts contained in PLIB use the Part Type Family Name. YOU apply the tags at the time of usage.

3. From a Logic Capture system. A Logic Capture system that creates a Connection list and a Parts list can usually generate ASCII files that can be edited into the format accepted by the PADS-2000 Program using the ASCII In command. This editing most likely will consist of (1) converting the format of the Connection list and Parts list to the PADS-2000 Program format, and (2) assigning the Part in the Parts list a PADS-2000 Program System Library Part Type name.

4. From the PADS-LOGIC schematic capture application program. PADS-LOGIC generates an ASCII IN file that is read directly into PADS-2000, using the ASCII IN input.

13.3 Manually Creating a Board Database Input ASCII File

This section of the Chapter describes how to manually create an ASCII file from a manually generated schematic. The format can also be used as background information to edit an output file generated by a Logic Capture system so that it can be entered into the PADS-2000 Program.

In the discussion which follows, the assumption is that a hand drawn schematic is the source and the Connection and Parts list is to be created in accordance with the format discussed below.

The minimum inputs to create a Board database are:

1. A Board Outline,
2. A Parts List,
3. A Connection List.

13.3.1 Board Outline

A necessary part of creating a Board Database is the Board Outline. While it is possible to manually code up a Board Outline using the corners of the Board Outline as reference points, this technique is not recommended. Instead, it is suggested that the Board Outline be created using the CREATE Editor (see Chapter 5) in a Binary or ASCII Job file and then recalled back into memory.

NOTE ON BOARD OUTLINE: A Board Outline created in the CREATE Editor is normally stored in the Parts Library LIB. When

created, the Board Outline file consists of Draw Lines but not Text. As long as the Board Outline does not contain Text, storing the Board Outline in LIB is proper. If, on the other hand, you wish to store Text as part of the Board Outline (examples: company name, "made in USA," etc.), then the proper storage technique is to create the Board Outline using the CREATE BOARD Editor, insert the Text as appropriate, and then store as a .JOB file using the JOB OUT command of the IN/OUT menu. The latter is the suggested sequence for designing a family of boards, all of which use the same Board Outline.

It is also recommended that the System Origin be co-located with the Board Origin and stored as part of the Board Outline.

The Board Outline Job file should be brought into memory and be displayed at the CRT at the time the ASCII Parts list and Connection Lists are read into memory using the ASCII IN command. The resulting database can then be saved as a Binary .JOB file.

13.3.2 Parts List

The ASCII Parts List is created using the PEDIT Editor (or other ASCII Editors). It is given a file name with the .ASC extension, and stored in the FILES subdirectory.

The following format is mandatory and is the minimum contents of a Parts List.

```
*PADS2000*
*Part*
*Standard*
AAA 1   BBB 1
AAA 2   BBB 2
  .
  .
  .
AAA N   BBB N
*End*
```

where

PADS2000 indicates PCB data to follow,

Part header indicates a Parts List to follow,
Standard turns on Standard Pwr & Gnd connections, if they are contained in the Part Type Information for the part.
AAA 1 is the Reference Designation of the first Part,
BBB 1 is the Part Type of the first Part,
AAA N is the Reference Designation of the last Part,
BBB N is the Part Type of the last Part,
End is the last entry or terminator of the Parts List.

The above is the basic content of a Parts List. Additional information that is optional can be contained in the Parts List as follows:

1. The addition of the Header *Standard* following the *Part* Header will automatically create a power (VCC) and ground (GND) connection NetList, using the Signal Pin (SIGPIN) data contained in the Part Type information for each part in the Part List. This command eliminates the laborious task of inputting the power and ground nets. Note, however, that if a part such as a connector or grounded spare gate of an IC, does not have this Power and Ground data contained in the Part Type data for the part, then these nodes will not be generated automatically and instead will need to be added to the Power and Ground nets by the designer.

2. A Part, such as a Connector part or Mounting Hole, can be prelocated on the board.

3. A Part can be reoriented or rotated from the orientation used during the construction of the Part Decal, and as stored in the Parts Library.

4. A part (such as a connector) can be glued down on the board.

The Parts List format utilizing both the mandatory and optional contents would be as follows:

AAA BBB XXX YYY ORI GLU

where
AAA is the Reference Designation,
BBB is the Part Type,
XXX is the X distance from the Board Origin to the Part Decal origin in thousandths of an inch,
YYY is the Y distance from the Board Origin to the Part Decal origin in thousandths of an inch,

ORI indicates the orientation in tenths of a degree from the orientation used in the original creation of the Part Decal,

where
0 equals the same orientation,
90.0 equals 90° counter clockwise rotation,
180.0 equals 180° counter clockwise rotation, etc.
GLU tells if the Part is to be glued down at the location defined:
U = Unglue, G = Glue, No entry = Unglue

13.3.2.1 Notes on Parts List Files:

1. The X and Y distance between the System board origin and Part origin is in 0.001" units. The assumption is made that the System Origin and the Board Origin are co-located; the parts will then be positioned in the proper location on the board relative to the Board Origin. If the System Origin is not co-located with the Board Origin, the X and Y placement of the Components will not be on the board.

2. After a Parts List is created in the PEDIT Editor, it is placed automatically in the *PADS-2000* Program subdirectory of \FILES for access by the ASCII IN command.

3. Use a file name of up to eight alphanumeric characters with the extension .ASC. The command of ASCII IN searches the \FILES subdirectory for the Parts List file name with the .ASC extension. If the file is not found, a

FILE NOT FOUND

message is issued.

4. As the ASCII Parts List file is read into memory, an error check is performed, and Error Messages, as appropriate, are output.

5. When creating a board database using the ASCII IN command, always read in the Parts List before the Connection List. The reason for this is that while the Connection List is being read in via the ASCII IN command, a checking process checks for the availability of each part called out in the Connection List by its Reference Designation. If

the Part Reference Designation is in the Connection List but not found in the Parts List which has previously been read in, a

PART NOT FOUND

message is issued. The error must be corrected before that line of the Connection List will be accepted.

13.3.2.2 Example Part List

The following is an example of a short Parts List where some Parts utilize both the mandatory and optional features, and other Parts use only the mandatory features:

```
*PADS2000*
*Part*
*Standard*
P1          CON26    15200    8500    0    G
U1          74LS00
U2          74LS04
U3          74LS00
U4          74LS00
U{5-20}     74LS04
C1          CL25
C2          CL25
C3          CL25
C{4-20}     CK05
R1          R1/4W,1K,5%
*End*
```

From the above Parts List, we see that connector P1 uses Library Part CON26, is located at X=15.2", Y=8.5" from the Origin, the Part is oriented the same as originally created and is glued down at the indicated position.

U1-20 are the IC's as indicated. They are not pre-located, are as per their original orientation and are free to be moved. A similar situation holds for the CAPS on the Parts List. The entries for U{5-20} and C{4-20} are interesting in that they provide a shorthand method of entering a long series of Part Reference Designations using the identical

part, without using an individual line for each part. The shorthand is interpreted as
follows:

First letter is the Reference Designation stem or family, i.e., U, C, etc.
next is the opening bracket {
next is the range of Part Reference Designations, i.e., 5-20
next is the closing bracket }
next is a space followed by the Part Type.

Example: U{5-20} 74LS04 is read:

U5 thru U20 use a 74LS04 Part Type.

Example: R1 R1/4W,1K 5%. This example uses the "Family" and "Tag" approach for identifying the Resistor R1. R1/4W is the "Family". R1/4W is the part type for the entire family of Quarter Watt Resistors. This Part Type name calls the Part Type from the library 1K 5% in the "Tag" which is tagged to the part after it is brought into the board database and uniquely identifies the resistors.

Notes on Parts List Format:

1. There is a maximum of six columns of mandatory and optional data.

2. No entry in a column is assumed to be the equivalent of a zero entry.

3. If the data of a column is to be accepted, all columns to the left of that column must have entries of data. In the instance of no data, you still would enter a 0.

Example: If, in the above Parts List, all IC's were to be glued down, "G" would, therefore, need to be entered in the GLU column for each IC. In addition, the X, Y and Orientation columns, columns to the left of the GLU column would require an entry of 0 for each IC in the Parts List.

13.3.2.3 Parts List Error Message

As the Parts List is read in through ASCII IN, it is checked for data inconsistencies. Any errors found will result in a hard copy printout of the error, (or the creation of an error file, if no hard copy). In addition, the offending line of data will not be accepted until corrected. Some Error Messages and corrective action are listed below.

Error Message:	No *PADS2000* found at beginning of file.
Correction:	Add correct Header.
Error Message:	Expecting a valid *XXX* command statement.
Correction:	Examine list. Some command statements starting with * is either in error or else missing altogether.
Error Message:	No *End* command found at end of file.
Correction:	Add *End* statement.
Error Message:	Bad *Parts* ASCII Data Line Format U11.1
Correction:	Something is wrong with the line containing U11.1

13.3.3 Connection List

The Connection List is created using the PEDIT Editor, given a File name with the extension .ASC and stored in the \PADS\\FILES\ subdirectory.

The following format is mandatory for a Connection List:

```
*PADS2000*
*Net*
*Signal*  Name 1  Width
Node 1  Node 2  Node 3
*Signal*  Name 2
Node 1  Node 2  Node 3
 "
 "
 "
*Signal*  Name 3
Node 1  Node 2  Node 3
*End*
```

where
PADS2000 and *Net* are headings for the Connection List.

Signal is the heading for each Net, and starts the line containing the name of each individual net.

Name is Name of net. A Net Name is required for each net.

Width is Track Width in units of the database. The Track Width remains the same for successive nets and does not require a new entry until the Net Track Width is changed.

Node is a Part Reference Designation and Pad/Pin Number.

13.3.3.1 Notes on Connection List

1. A To-From connection represents the Electrical connection between two or more physical nodes (part pin/pads).

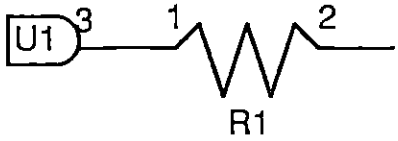
2. A node consists of two items of data: (1) the Part Reference Designation, i.e., U1, R1, etc., and (2) the Part pad/pin number.

3. Reference Designation Format
This entry is used to give a name to a Part. Up to six alphanumeric characters can be used, however, the first character must be an alpha character. More than six characters cannot be used. Non-alphanumeric characters such as -, / , (, space, etc., are not acceptable.

Acceptable	Not Acceptable
U1	U1-C
C101	C5000/8
RP10	2R3

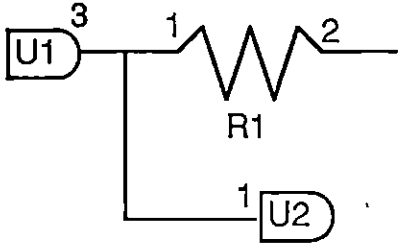
4. A space is used to separate two different nodes; within a node, a period is used to separate the Reference Designation and the pad number.

1) A Single Tree Connection



is represented by U1.3 R1.1
Note the space between the two nodes.

2) A Multi-Tree Connection



is represented by U1.3 R1.1 U2.1

5. *Signal* Statements and To-From Signal Names

A *Signal* statement and Name is used for two purposes: (1) to identify different (Electrically) To-From trees, and (2) to assign a signal name for a To-From tree. The signal name is from the Logic diagram, i.e., SIGA, CLK, +5V, GND, etc. If unnamed, you must still use the *Sig statement to identify the connection and you must supply the name.

Example:
Signal D0
U1.3 R1.1 U2.1 A 3-Node To-From Connection Tree named D0

NOTE: Signal Names must obey the following rules:
1. Maximum character length 12.
2. Spaces or blanks are not legal.
3. Letters and numbers can be mixed.

4. The following are illegal:

```
^ [ ] &
*
{ } ( )
@  ? = \
```

Remember, there must be a *Signal* statement between each successive To-From tree/net. If there is not a *Signal* statement, the two trees will be tied together!

6. Discrete Component Pad Numbering

Discrete components normally are not pin numbered on the Logic Diagram. They must, however, have a pin number in order to be entered into the database.

7. Tying Nets Together

In Logic Diagram documentation, parts of a multi-node connection tree can be depicted anywhere--on all sheets and at various locations on a sheet, etc. In preparing your To-From list, you do not need to enter the entire tree as a single continuous entry. You can enter the different parts of the common connection at different locations of the To-From list, and the PADS-2000 system will tie the partial nets together, providing you use a common signal name for each partial connection.

Repeating a signal name at different locations within a connection list will result in all entries in the To-From list that use the same signal name, being tied together.

```
Example:  *Signal* D0
          U1.3   R2.1   U1.6
          *Signal* D0
          U5.2   R6.1
          *Signal* D0
          U5.8

results in  *Signal* D0
          U1.3   R2.1   U1.6   U5.2   R6.1 U5.8
```

Use the "Common Signal-Name" technique (above) to enter a very long multi-tree connection that requires several lines of data in the To-From list. These different lines of data may be scattered throughout the list. Simply use a Signal Name that is common on all lines of data. This common signal name will tie all lines of data into a long multi-node connection tree.

8. Abbreviated Notations

Certain type connection trees which have a common pin, such as Power trees, Ground trees, and Memory trees can use an abbreviated or shorthand format for entering the tree. This abbreviated format uses the format X{F-L}.P, where X = the Reference Designation letter, i.e., U, C, etc., F = the first Reference Designation number of a continuous series of Reference Designations, L = the last Reference Designation number in the series, { } is the abbreviated notation symbol and P is the common pin/pad number. Note that a - separates the F and L and a . precedes the P.

Example: *Signal* GND 50
U{1-50}.7

results in a common tree for GND, which is read U1 through U50, pin 7. The system will expand the abbreviated format into the full GND connection tree.

9. Width

The Width entry is in thousandths of an inch, i.e., 12, 50, etc. The first entry of a net establishes the width of all succeeding nets until a different width is entered. It is suggested, therefore, you arrange the Connection lists by Width, i.e., Signal, Ground and then Power.

10. Suggested Forms

Use the following suggested form for your To-From data and Component List when transferring from a manually created schematic for entry into the ASCII Connection List.

REF DES	PAD	REF DES	PAD	REF DES	PAD	REF DES	PAD

13.3.3.2 Example of a Connection List

```
*PADS2000*
*Net*
*Signal*    D0    12
P1.1  U1.1  U2.3  U4.5  U6.1  U7.2
*Sig    D1
U1.2  U2.4  U3.3
*Signal*    D2
U1.6  U2.5  U3.4
*Signal*    GND    50
U1.7  U2.7  U3.7  U4.7  P1.7
U8.7  U9.7  U10.7
*Signal*    PWR    50
U1.14  U2.14  U3.14  U4.14  P1.14
U8.14  U9.14  U10.14
*End*
```

NOTE: The automatically generated power and ground connection list, generated from the SIGPIN data contained in the Part Type information of the Parts Library will NOT contain power and ground nodes not identified in the Part Type data. Examples of this are power and ground pins of an I/O connector, the grounding of spare gates of an IC, etc. This data will have to be entered in the power and ground net by the designer.

13.3.4 Connection List Error Messages

As the Connection List ASCII file is read into the system and merged with the Parts List, certain checks are made on the data. Errors found

F5 Delete Line. When selected, deletes the line at the current location of the cursor.

F6 Undelete Line. When selected, recovers the LAST line deleted. Warning: Recovers one deleted line only.

F7, F8 Begin/End of line. Moves cursor to Begin or End of line, where cursor is located.

F9 Quit. Preparatory command for leaving the Editor WITHOUT, repeat WITHOUT saving the file being edited. When selected, results in the message:

CONFIRM QUIT (Y/N)?

F10 Write. Used for writing the Editor file to storage. When selected, results in the message WRITE FILE NAME? Carriage return will result in the file being written to storage, using the name by which the file was originally called into the editor. A back-up file using the name with the extension .BAK is also created. Otherwise, if desired, enter a new or different name.

The cursor movement is controlled by the Home (top of file), End (bottom of file), Right, Left, Up, Down arrows, Page Up and Page Down keys.

CHAPTER 14 - FILES MANAGEMENT

14.1 Objective

This chapter discusses the control and storage of the files created with the system. It is absolutely essential that you understand and follow the practices and procedures described below. Failure to do so will result in lost data, in which you have invested a considerable amount of your time.

14.2 Job Directory Listings

As you have been storing design Jobs either in Binary or in ASCII, they have been written on the hard disk in a subdirectory of PADS called FILES. From time to time, you will want to know what design files are stored, whether or not a specific design is on your hard disk, how much additional remaining space you have on your hard disk, etc. To do this, you will need to obtain a printout or CRT listing of the subdirectory FILES.

To obtain a listing of all files in the FILES subdirectory, proceed as follows:

To the display: **C:\>**

Enter: **CD\PADS\FILES**

The Response: **C:\PADS\FILES** indicates that you are now in the subdirectory FILES containing all Job files. To the Response: **C:\PADS\FILES>**

Enter: **DIR <CR>** for Directory.

A listing of the contents of the FILES subdirectory will appear at the CRT, along with the date of the file, size of each file, number of files and free space remaining on the hard disk. It is a good practice to

examine your FILES subdirectory contents on a daily basis and delete those files that have outlived their usefulness. Before deleting a file however, be sure you will never need the file!

14.3 Job/Disk Management and Control Procedure

You have recognized that in addition to containing your PADS-2000 Program, your hard disk will also contain an ever growing number of both Binary job files and ASCII design files. In this section, we will discuss the Control and Management of these files so that the loss of valuable data is minimized and you know where your designs are at all times.

During the process of designing a board, we would suggest that in addition to the Automatic Backup made every few minutes by the system, you should make a Binary Job Save approximately every hour, or at the beginning and completion of each significant event, i.e., immediately after the reading in of the initial ASCII file, the starting of Placement, the completion of Placement, starting of Routing, every hour or so during Routing, the completion of Routing, starting of Checking, and a final file, upon completion of the design and ready to Post Process. This final file becomes the Archival file.

This may sound like a lot of files to make but, remember, disk space is cheap as compared to your time; in the process of designing a board you can probably make in the order of 50 or more board saves on your hard disk, and all but one of these files are considered temporary and will eventually be deleted before you start your next board. (You will save the final file as the Archival file).

To summarize, the general philosophy is to be liberal with your Saving of Binary JOB files. In this fashion, if something should happen to your work in process, i.e., it becomes contaminated or you have designed the board into a "blind alley," then you do not have to go back to the beginning and start all over again. You simply go backwards in the stored job sequence until you find the appropriate job file and start again from that point. In this regard, it is important to keep a written Job Save Log during the design of a board in order to assist you in identifying the contents of each Job Save file.

After you have made 20 or 25 Job Saves on a board, it is difficult to remember the contents of each Save. Therefore, as your job design progresses, we suggest that you keep a written Job Save Log similar to that shown in Figure 14.1 below:

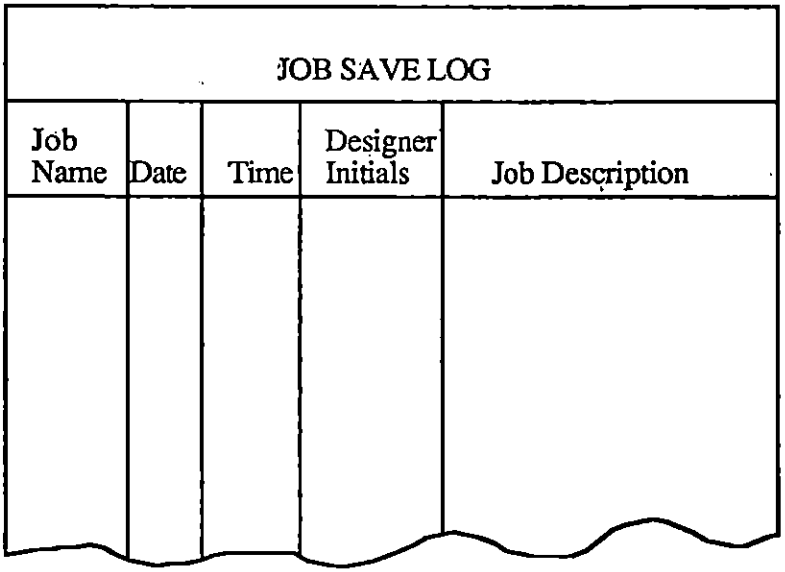


FIGURE 14.1 - SUGGESTED JOB SAVE LOG

14.4 Deletion of Files

After the board is designed and post processed, we strongly recommend that you remove all of the interim files created during the process of design that are on the hard disk, leaving only the last or completed file for use as the Archival file. This is accomplished as follows:

First, just in the event you make an error during the deletion of interim files. PLAY IT SAFE by copying ALL files associated with the board on the hard disk in the FILES Directory to a floppy disk. Keep this floppy until the deletion of all unnecessary files on the board is complete, for use in restoring the hard disk, if necessary.

To delete files from the FILES subdirectory, use the DOS command of DEL or Delete. Be in the PADS\FILES path and enter the Delete command along with the Job Name and Extension:

C:\PADS\FILES>DEL XYZ.JOB <CR>

HINT 1: Before deleting any files, make a hard copy Directory listing of FILES. Check off only those you want to delete. Double check your decision! After you have deleted the interim files, make a listing of the directory to insure that you have deleted what you intended to delete and have left on the disk, those files that you wish to keep.

14.5 Daily Backup

We recommend that at the end of your shift, you always save the last Job file created that day. This is an "insurance" policy against a system failure or human oversight that deletes, overwrites or otherwise destroys your effort that exists on hard disk. Should such an unhappy event occur, you have your floppy disk of your work, which you load back onto hard disk and proceed.

14.5.1 Making a Backup Floppy

To make a Backup, access the FILES subdirectory by entering:

CD\PADS\FILES.

Use the following command in response to C:\PADS\FILES.

Copy C:XXX.JOB A:XXX.JOB <CR>

The correct translation of the above is: You are currently in the subdirectory PADS\FILES on the hard disk. You wish to copy from this subdirectory a file called XXX.JOB. You want to copy the file to the floppy disk contained in Drive A and you wish it to be named XXX.JOB. The disk light will light up while the transfer is occurring and the message:

1 FILE COPIED

will occur. You should at this time obtain a directory of the floppy disk in drive A by entering the command

DIR A:<CR>

to insure that the file XXX.JOB has been placed on the floppy.

In the event there is not sufficient room on the floppy disk to copy all the files, use a second disk and copy the files not copied on the first disk.

After you have copied all your files created during the day onto a backup floppy disk, make a print out of the Directory of the contents of the floppy , and label the disk, and protect this disk.

14.5.2 Using the Backup Disk

Let's say you come to work the next day and you find that half of the files you had created on hard disk, the day before have been erased by the night shift designer. How do you get your backup file on floppy disk back on the hard disk?

Enter the PADS\FILES subdirectory such that the display is C:\PADS\FILES> and have the backup disk in drive A. You enter:

Copy A:*.* <CR>

This will copy ALL the files on the floppy over to the hard disk and place them in \PADS\FILES. Any existing files on the Hard Disk that are also on the floppy disk will be overwritten and any missing files on the Hard Disk will be copied from the floppy disk, such that at the end of the transfer, all files on the floppy disk will now be on the hard disk. Use DIR to check.

Alternatively, you could have copied only the erased files.

14.6 Archival Files

The file on the hard disk representing the finished board and the file that was used for post processing artwork should be considered the Master or Archival file and placed in your company's archives. It should be controlled with exactly the same care your company's Configuration Management and Documentation Control system controls a Master drawing. To do this, we suggest you use a floppy disk technique if you do not have a Mag tape on your system or access to a Mag tape. Use only high quality, High Density floppy disks.

Copy the final file used for post processing on to two different floppy disks: Archival Disk 1 and Archival Disk 2. Label them accordingly. We suggest you do not put jobs from other boards on these disks. Dedicate a disk to the files associated with one board, including its revisions. Disks are inexpensive and the "one board--one disk" procedure will greatly simplify your administration. Store the two disks safely in two separate locations.

The two separate locations should be in two different buildings or sufficiently separated so that if a catastrophe such as a fire destroys one set of Archives, it does not destroy the other.

At some time in the future when a revision is made to the particular board, copy the database from the archival floppy, read it into the hard disk and make the revision, taking care to uprev the file name, i.e., XXXA.JOB, XXXB.JOB, etc. Store the revised file on the same two archival floppies where the original job was stored. In this fashion, you will be able to dedicate a floppy disk to a single board and all of its revisions, thus greatly simplifying your record keeping and retrieval of work that you have performed, yet providing protection to your files.

The DOS copy command for copying from hard disk to floppy disk are the same as for backing up your work files.

When you have successfully copied your Archival file from hard disk to a floppy (obtain a listing of the floppy disk, insuring the file has been copied, then read the file back into the hard disk [using a different name] and check that it can be called up in PADS-2000, you can then delete the Archival file and the Check File from your hard disk, thus freeing up more hard disk space.

A NOTE OF CAUTION: Occasionally, in the process of transferring from hard disk to floppy, a data error may occur, thus invalidating the copied disk. To guard against this, each time you transfer to the floppy, immediately transfer it back to the hard disk and attempt to call it up using the JOB IN subroutine. In this fashion, you can be assured that the database for your completed board has been successfully transferred to the floppy and can be retrieved at some time in the future.

We can see that if you are designing a 1000 IC board, its database would exceed the capacity of a High Density Floppy Disk. In such an instance, have a second disk available and use the DOS command of BACKUP. In the process, use the DOS command of VERIFY to maximize the error checking during the Disk Write application.

In the event that you have access to a Mag tape on your system, you may want to use this media as your Archival medium rather than floppy disks. In this case, follow the same procedures of making two different Archival tapes, reading them back into the system and bringing them up at the CRT to insure that the copy has been made successfully and store the two mag tapes in separate locations.

14.7 Storage Space

You should be aware of the storage space required for the database of a board, together with the capacity of a floppy disk.

The PADS-2000 database requires 4 KBytes of disk storage for each equivalent 14 pin DIP on the board. Example: a board with 100 equivalent 14 pin ICs would require approximately 400 KBytes of storage.

14.8 Software Updates

From time-to-time, you may receive more recent versions of the PADS-2000 software. The board files created with your older software can be ported to the new software and manipulated if you observe the following:

1. Binary files (with Extension.JOB) may or may not be portable to the newer, higher revision software, depending upon the old revision

version. Usually, files made in a the previous version will be upwards compatible, but files that were made on very old versions of PADS-2000, or on PADS-PCB 2.0 and earlier, will not be compatible. Try to use the Job IN command to read the file. If not successful, make an ASCII Out File using the old software release, and then read the file into the new PADS-2000, using the ASCII IN command.

2. ASCII files are always portable to the higher revision software. Accordingly, using the older software, convert the board database into an ASCII OUT file. Read the ASCII file into the New Software where the board file can be manipulated and stored.

INDEX

A

Absolute Versus Relative Coordinates 85
Absolute/Relative Coordinate Mode 89
ADD CONN 50
ADD LVL 68
ADD PART 54
ADD ROUTE 57
ADD RUT 222
Adding a Connection 50
Adding Arcs in a Board Outline 111
Adding Text to the Decal 128
Alternate Parts 140
Annular pad shapes 67
Archival Files 320
ASCII 32
ASCII File 25, 30, 31
ASCII files 26, 297, 322
ASCII Format 3
ASCII IN 30
ASCII In file 297
ASCII OUT 32, 34
ASCII OUT File 310
Auto Macro 86
Auto Path Router 200
Auto Rename of Components 162
Auto Steer Router 201
Autoplacement 144
Autoplacement of Discrete Parts 154
Autorouting with PADS-2000 178

B

Backup 29
Backup job 28
Backups 77
BakGnd 61
Batching CAM Outputs 250

Binary Job 25
Blocking Routing Channels 185
Board Edge Clearance 98

C

CAM 3
CAM Outputs 233
CHANGE TEXT SIZE 103
Changing a Part Type 220
Changing Tracks Globally 71
Check ASCII 215
Check Future Net 216
Check Plane 217
Check Schema 216
Checking 2, 213
Color Indicator 61
Color Selection 60
Combining 2-D Lines and Text 123
Component Placement by Group 159
Component Side Convention 11
Connection Density Histogram 157
Connection Density Map 158
Connection Hot Spots 184
Connection List 304
Connections 62
Control Attributes 79
CONV IN 44
Copper Areas 113
Copper in Decals 127
Copper Pour Clearance 77
Copper Share 83
Copying 2-D Items 121
Copying a Copper Area 116
Copying a Text Item 104
Create 2, 101
Creating a Board Outline 105
Creating a 2-D Line 120
Creating a Board Outline 101
Creating a Circular 2-D Item 120
Creating a Copper Pour polygon 101
Creating a Macro 84

Creating a Part Type 101
Creating Circular Board Outlines 106
Creating copper areas 101
Creating Cut Outs in Poured Copper 118
Creating general Text 101
Creating Non-Circular Board Outlines 105
Creating Part Decals 101, 124
Creating Placement Matrices 144
CTRL-I 13
Cursor 10
Cursor Location Indicator 12

D

Daily Backup 318
Default Track Width 98
Defining a Group 159
Defining Pad Stack Sizes 63
DEL CONN 51
DEL LVL 68
DELETE PART 55
Deleting 2-D Items 122
Deleting a Board Outline 112
Deleting a Connection 51
Deleting Text 104
Deletion of Files 317
Density 186
Design Rule Check 213
Design Rule Checking 77
Determining the Length of a Track 198
Determining the Minimum Net Length 155
Dimensional units 33, 59
DISCONNECT PIN 53
Discrete Part Naming Convention 294
Display 60
Display Dot Grid 132
DISPLAY LEVEL 23
Display Mode 75
Display of Nets 81
Dot grid 11, 59, 76
Drill 65
Drill 235

RENAME NET 52
RENAME PART 56
REPORTS 225
Reports 3
Resolution 11
Rip Up 83
ROT 90 103
Rotating Components 141
Routing Grid 98, 180
Routing with On-Line DRC 202

S

SAMPLE FUTURE NET FILE 38
Schema In 48
SEARCH 21
Selecting Tear Drop Pads 211
Selecting Visible Nets 179
SET ANG 103
Setting the Units 132
SetUp 2, 59
Shape 65
Shove Router 199
Shove Rules 83
Show Level 73
Size 65
Size of Text 69
Software Updates 321
Solder Side Convention 11
Spacing 59, 214
SPECIFY CAM OUTPUT SUBDIRECTORY 258
SPIN 103
STARTUPL.ASC File 311
Statistics 227
Storage Space 321
Storing a Design 28
STRAT IN 203
STRAT OUT 204
Swap Parts 149
Swapping Gates 151
Swapping Pins 152
System Parameters 72

System Parts Library 292
System Units 96

T

T-Routing 172
T-Routing feature 58
Tear Drops 74, 75
TERMINALS 129
Text 14
Text Command 102
Text Size 59
The Engineering Changes (ECO) Menu 3
The Input/Output Menu 2
The Power and Ground Router 187
TOLERANCE 39
Top Menu 89
Track Line Width 176
Track Routing 2, 165
Transferring library items 277

U

Unused Report 226

V

VALUE 39
Verify 257
VERT 82
Via Shift Allowed 98
VIA TYPE 22
Viewing Window 17, 19
Virtual memory 5

W

Wet Ink Plots 233
Wet Ink Quality Plots 259
Width 14, 50
Width of Non-Poured Copper Areas 71
Width of the 2-D Lines 71

PADS-386 SUPERROUTER

TABLE OF CONTENTS

CHAPTER 1
INTRODUCTION TO PADS-386 SUPERROUTER 1

CHAPTER 2
TESTING THE INSTALLATION 3

CHAPTER 3
PADS-386 SUPERROUTER FEATURES 9

CHAPTER 4
CONTROLS FOR PADS-386 SUPERROUTER 11
4.1 ENTERING THE PADS-386 SUPERROUTER 11
4.2 LOADING THE BOARD DATABASE 12
4.3 SET WIDTH (F2) 16
4.4 SELECTING THE DIRECTION RULES 17
4.5 SETTING THE ROUTER STRATEGY 17
4.6 ROUTING GRID 19
4.7 TRACK, CLEARANCE AND PAD SIZES 21
4.8 REFERENCE TABLES 22
4.9 OFF GRID PADS 24
4.10 Use of Copper 26
4.11 TREATMENT OF TEXT AND 2-D LINE ITEMS 26
4.12 SETTING THE ROUTED LAYERS 27
4.13 ROUTING MIXED WIDTH NETS 27
4.14 DEFINING CONNECTIONS NOT TO BE 28
4.15 PARTIALLY ROUTED TRACKS 28
4.16 FIXED PREROUTED TRACKS 29
4.17 SMD EXIT VIAS 30
4.18 SETTING UP THE PADS DATABASE 30
4.18.1.1 SETUP, SR PARAMETERS 31
4.18.1.2 SETUP, NET ATTRIBUTES 33
4.18.2 THE SETUP FILE FOR PADS-PCB 35
4.18.2.1 CREATING A SETUP FILE 36
4.18.2.2 LAYER ROUTING DIRECTION 36
4.18.2.3 CORNER TYPE 37
4.18.2.4 FIXED TRACKS 37

4.18.2.5 PARTIAL TRACKS 38
4.18.2.6 FIXED CONNECTIONS 39
4.18.2.7 BOARD OUTLINE 39
4.19 ROUTING AND PLACEMENT GRID SYNC 40
4.20 RETURNING A ROUTED FILE TO PADS 42

CHAPTER 5
DESIGNING WITH SURFACE MOUNTED DEVICES 45
5.1 PREPARATION OF THE DATABASE IN PADS- 45
5.2 TRANSFER TO PADS-386 SUPERROUTER 46
5.3 PADS-2000 47
5.4 ROUTING OF SIGNALS IN SUPERROUTER 48
5.5 PREPARATION OF THE DATABASE IN PADS-PCB 49
5.6 TRANSFER TO PADS-386 SUPERROUTER 49
5.7 PREPARATION FOR SIGNAL ROUTING IN PADS-PCB 50

CHAPTER 6
ROUTING WITH PADS-386 SUPERROUTER 51
6.1 THE ROUT ALL OPTION 51
6.2 THE INITIAL ROUTER 52
6.3 ROUT RIPUP (THE RIPUP ROUTER) 53
6.4 THE OPTIMIZER ROUTER 54
6.5 INFORMATION DISPLAY 55
6.6 INTERRUPTING THE SUPERROUTER 56
6.7 SUPERROUTER REPORTS FILE 56

CHAPTER 7
ROUTING CHECKLIST 57
7.1 PADS-2000 CHECKLIST 57
7.2 PADS-386 SUPERROUTER CHECKLIST 59

CHAPTER 8
STRATEGY OPTIONS 61

CHAPTER 9
USING THE PADS-386 SUPERROUTER 65
9.1 ESTIMATING ROUTING TIME 65
9.2 TESTING ROUTABILITY 67
9.3 GETTING THE MOST FROM PADS-2000 68

CHAPTER 10
MEMORY REQUIREMENTS 73

CHAPTER 11
ERRORS AND TROUBLESHOOTING 75
11.1 ERROR MESSAGES 75
11.2 TROUBLESHOOTING 78

CHAPTER 12
SUPERROUTER COMMANDS 79
12.1 COMMAND NAME: INPUT 80
12.2 COMMAND NAME: ROUT 81
12.3 COMMAND NAME: EXIT 82
12.4 COMMAND NAME: STRATEGY 83
12.5 COMMAND NAME: REDRAW 84
12.6 COMMAND NAME: READ 85
12.7 COMMAND NAME: SET WIDTH 87
12.8 COMMAND NAME: WORK DIR 88
12.9 COMMAND NAME: LAYER ASGN 89
12.10 COMMAND NAME: COLOR ASGN 90
12.11 COMMAND NAME: READ STOP 92
12.12 COMMAND NAME: ROUT ALL 93
12.13 COMMAND NAME: ROUT INIT 94
12.14 COMMAND NAME: ROUT RIPUP 96
12.15 COMMAND NAME: ROUT OPT 97
12.16 COMMAND NAME: ROUT STOP 98
12.17 COMMAND NAME: ZOOM 99
12.18 COMMAND NAME: OPT SPEED 100
12.19 COMMAND NAME: STRATEGY 101
12.20 COMMAND NAME: LOAD STRAT 102
12.21 COMMAND NAME: SAVE STRAT 103

INDEX

CHAPTER 1

INTRODUCTION TO PADS-386 SUPERROUTER

The PADS-386 SUPERROUTER is the advanced Autorouter from CAD Software, Inc. It is based on the successful Ripup and Retry autorouting approach, and is capable of achieving 100% results on most printed circuit layouts.

PADS-386 SUPERROUTER functions with both PADS-PCB and PADS-2000 as host CAD systems. Unrouted boards from PADS-PCB can be routed and the routed board returned to either PADS-PCB or PADS-2000. Unrouted boards from PADS-2000 can be routed and the routed board returned to PADS-2000 only.

When PADS-386 SUPERROUTER is used with PADS-2000, it supports routing grids of 10, 12.5, 16.67, 20, 25 and 50 mils. When used with PADS-PCB, routing grids of 10, 12, 16, 20, 15 and 50 are supported.

Up to 12 routing layers may be specified, and simultaneously routed when used with either PADS-PCB or PADS-2000.

PADS-386 SUPERROUTER is useful for a wide variety of design technologies, from very dense digital designs, to multi-layer SMT designs, to predominantly Analog layouts.

Routing success depends on the proper control of the router. YOU SHOULD REVIEW THIS APPENDIX, IN DETAIL, TO LEARN HOW TO EFFECTIVELY USE SUPERROUTER TO ROUTE YOUR DESIGNS.

PADS-386 SUPERROUTER is designed as a Stand-alone Program Module to PADS-PCB and PADS-2000. Use of these programs is described in this appendix.

The remainder of this section describes the organization of this manual.

CHAPTER 2, Testing the Installation, describes a test that should be conducted to insure the program is installed correctly.

CHAPTER 3, contains a "Quick-look" Summary of the features you can expect with PADS-386 SUPERROUTER.

CHAPTER 4, Controls for PADS-386 SUPERROUTER, discusses the use of the SUPERROUTER software.

CHAPTER 5, Designing with Surface Mounted Devices, discusses a special feature, the Automatic creation of "Exit Vias" that enhances the usefulness of the PADS-386 SUPERROUTER.

CHAPTER 6, Routing with the PADS-386 SUPERROUTER, describes the various routing passes including ROUTE ALL, the INITIAL Pass, the RIPUP and REROUTE Pass and the MANUFACTURING OPTIMIZATION Pass

CHAPTER 7, contains the Checklist for both PADS-2000 and PADS-386 SUPERROUTER activities.

CHAPTER 8, Strategy, describes the concept of the Routing Strategy, and how Routing Strategy is established.

CHAPTER 9, Using the SUPERROUTER, provides Hints as to how to judge if the PADS-386 SUPERROUTER is going to be successful on its board, and how you can increase its effectiveness.

CHAPTER 10, Memory Requirements, specifies the Memory Requirements. Read this section carefully to establish the memory requirements for your boards.

CHAPTER 11, Error Messages & Troubleshooting, provides a listing of error messages and troubleshooting procedures.

CHAPTER 12, SUPERROUTER Commands, provides detailed information about the menu commands in the PADS-386 SUPER ROUTER.

CHAPTER 2

TESTING THE INSTALLATION

This chapter of the manual addresses the installation and testing of your PADS-386 SUPERROUTER Installation. Your PADS-386 SUPER ROUTER package contains the following:

- One floppy disk containing the PADS-386 SUPERROUTER software and test files .

- A security key programmed for PADS-386 SUPERROUTER (may also contain either PADS-2000 or PADS-PCB), a PINSTALL manual containing the step-by-step instructions for installing the PADS-386 SUPERROUTER (also for installing other PADS products).

During the installation of the PADS-386 SUPERROUTER, take care to follow exactly, the instructions contained in the PINSTALL Manual.

If, at any time, you have questions regarding the installation process, call CAD Software, Inc. (508) 486-3328, and ask for Technical Support on your PADS-386 SUPERROUTER Installation. Our Support Staff will be pleased to assist you.

INSTALLATION TESTING

This Installation Test is applicable to either PADS-PCB or PADS-2000 when used with the PADS-386 SUPERROUTER.

Following the installation of the PADS-386 SUPERROUTER Software, you SHOULD route the Test exercise included with the software. This Test Exercise consists of the transfer of a test file called RUTTEST from either PADS-2000 or PADS-PCB to SUPER ROUTER, the routing of the board and transfer back to the Host Design system. By completing this exercise, you will have confidence that the PADS-386 SUPERROUTER has been installed properly, and is performing correctly.

Moreover, you will be introduced to the PADS-SUPERROUTER and can observe the sequence of Routing at the CRT.

Proceed as follows for the Test:

Have either the PADS-2000 Shell or PADS-PCB Shell displayed at the CRT. Using the mouse cursor, select the SUPERROUTER Port (box) as displayed in the Shell.

A Text screen is shown at the CRT, indicating the current version of PADS-386 SUPERROUTER.

When the SUPERROUTER program is executed, the CRT Display changes to a "PADS-type" presentation identifying the Version Number (X-XX), followed by the Copyright Notice.

The command menu is shown in the following Figure 2.1

1 Input
2 Rout
3
4 Strategy
5 Zoom In
6 Zoom Out
7 Save Job
8 Redraw
9 Change Job
0 Exit

Figure 2.1 - SUPERROUTER MAIN MENU

Select INPUT (F1) using the Function Keys.

NOTE: The Function Keys, or the Mouse Cursor, are used to select menu selections in PADS-386 SUPERROUTER.

The CRT presentation changes to the INPUT screen, as shown in the following Figure 2.2, "INPUT MENU."

1 Read
2 Read Rout
3 Set Width
4 Work Dir
5

6 Layer Asgn
7 Color Asgn
8
9 Read Stop
0 Exit

Figure 2.2 - INPUT MENU

Select READ (F1). In response to the prompt:

Input file name: \ PADS \ Files:

Respond:

RUTTEST <CR>

In response to the prompt:

Delete existing report file (Y/N)

Respond:

Y <CR>

In response to the prompt:

Enter memory allocation for PCB database:

Type:

<CR>

In response to the prompt:

Pause if messages fill screen:

Type:

Y <CR>

In response to the prompt:

Enter plane net names [max 12] *(PADS-PCB only)*

Type:

<CR>

In response to the prompt:

Routing Grid [25]?

Type:

<CR>

In response to the prompt:

Min Clearance [13]?

Type:

<CR>

In response to the prompt:

Default Track Width [12]?

Type:

<CR>

The file RUTTEST will be read into the PADS-386 SUPERROUTER program, and a series of messages showing the progress will be displayed. At completion, the Test file will be displayed at the CRT (a small test board), and the message:

File Read Done

is displayed.

You will now test the SUPERROUTER by routing the RUTTEST file.

Exit the INPUT menu, by selecting ROUT (F2). The menu will change to that of Figure 2.3

1 Rout All
2 Rout Init
3 Rout Ripup
4 Rout Opt
5

6
7
8 Opt Speed
9 Rout Stop
0 Exit

Figure 2.3 - ROUTER MAIN MENU

Select ROUT ALL (F1). This will start the PADS-386 SUPER ROUTER. As it routes, you will see the results displayed, numerically in the Status Information window, and graphically, at the CRT. This small demonstration test board will route 100% in a few minutes, depending on the speed of your computer. All three phases, INITIAL, RIPUP, and OPTIMIZE will be used. When routing is completed, you are asked to save the routed connections to a Job File.

Upon completion, in response to the prompt:

Save routed connections to Job File:

Type:

Y <CR>

In response to the prompt:

Update original Job File [Y/N]

Type:

N <CR>

In response to the prompt:

Input new Job File:

Type:

Your Initials <CR>

In response to the prompt:

Delete Router Report file [Y/N]:

Type:

N <CR>

In response to the prompt:

Exit to DOS or Reload SUPERROUTE [Exit/Reload]

Type:

E <CR>

You will be placed in the PADS Shell. Select PADS-2000 or PADS-PCB, whichever Host design system is being used. When the PADS program is initialized, use IN/OUT (F1) to bring in the routed file using your initials as its name.

The Routed file will be displayed in the PADS Host system and, thereafter, can be handled as a PADS .JOB File.

NOTE: If the PADS-386 SUPERROUTER program performed as described above, you can assume that your hardware and software are operating satisfactorily, and your PADS-386 SUPERROUTER software has been successfully installed. You may, at this point, proceed with your study of the PADS-386 SUPERROUTER.

A suggestion: After the Installation test using RUTTEST has been successfully completed, rerun the test 2-3 times to give you a sense of the flow from PADS to SUPERROUTER, of the routing efforts, and the transfer back to the Host. Moreover, as you progress through the different features described in the following chapters, use RUTTEST to become familiar with the feature. RUTTEST is small and thus an ideal candidate to use to become familiar with SUPERROUTER in a short period of time.

CHAPTER 3

PADS-386 SUPERROUTER FEATURES

This chapter provides a "Quick-look" Summary of the PADS-386 SUPERROUTER Software and its features. This "Quick-look" is intended to give the user a rapid overview of SUPERROUTER, and what to expect, prior to becoming involved in the details of using the SUPERROUTER.

1) The PADS-386 SUPERROUTER works with either PADS-2000 or PADS-PCB. A board has its components placed in PADS-2000 or PADS-PCB; then it is moved to the PADS-386 SUPERROUTER program, routed and returned to PADS-2000 or PADS-PCB for post-processing.

2) In PADS-386 SUPERROUTER, there are three routing phases:

1) INITIAL ROUTE: A Single Pass Router pass that attempts to route as many connections, as possible. If 95% or more of the total connections are routed during INITIAL ROUTE, then the following phase (RIPUP and REROUTE) will probably route 100% of the connections. If the INITIAL ROUTE is significantly less than 95%, the RIPUP phase will probably not successfully complete 100%. In fact, the board is probably not routable, using the current design criteria.

2) RIPUP AND RE-ROUTE: A Router Pass that rips up the blocking tracks routed during the initial routing pass, and re-routes them, in an attempt to route the unrouted connections remaining from the INITIAL Router.

3) OPTIMIZATION: A pass that reroutes the completed board, in order to improve the manufacturability of the board. Tracks are straightened and spread evenly over the board area and vias are minimized. Two options are available: "Quick," but less thorough, or a more lengthy option, but very thorough. The Designer may select a single or multiple optimization passes--generally two fast or one slow are adequate.

3) These three routing phases may be run independently, or they may be all run sequentially, using the ROUTE ALL Command. ROUTE ALL, however, is the recommended method of using the three routing phases. The individual phases should only be used for testing, experimenting, etc.

4) Critical tracks may be completely routed, or partially routed in PADS-2000 or PADS-PCB, and "FIXED" prior to being routed in PADS-SUPERROUTER. These "FIXED" tracks will not be disturbed, (routed, ripped up or optimized) by PADS-386 SUPERROUTER, and are left exactly as originally routed in the Host Design system.

5) The PADS-386 SUPERROUTER STRATEGY feature has 19 different options that may be controlled by the Board Designer. These 19 options control cost parameters that influence the SUPERROUTER during the three Routing phases. Different "STRATEGY" files can be created, saved and re-used for similar board families.

6) Routing Grids are 10, 12.5, 16.67, 20, 25 and 50 mils for PADS-2000 database and 10, 12, 16, 20, 25 and 50 mils for PADS-PCB.

7) The Router observes the track size, air gap, and the PAD size and shape, as set up in PADS-2000 or PADS-PCB.

8) The direction of Routing on a layer may be controlled.

9) SMD Power and Ground connections will have automatically generated "Exit Vias" to internal plane layers, during Routing.

10) Up to 12 Layers can be routed by SUPERROUTER; all layers are routed simultaneously.

11) The CRT presentation may be expanded and contracted or repositioned, using the mouse cursor.

CHAPTER 4

CONTROLS FOR PADS-386 SUPERROUTER

This chapter describes the various controls used to run the PADS-386 SUPERROUTER Software.

4.1 ENTERING THE PADS-386 SUPERROUTER

PADS-386 SUPERROUTER is entered and exited from the PADS-PCB and PADS-2000 Shell in a similar manner.

The PADS-PCB Shell is as follows:

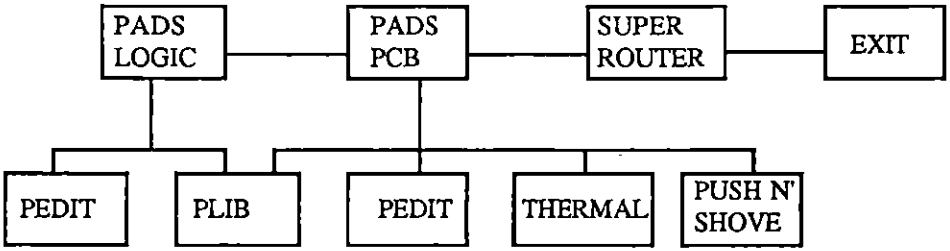


Figure 4.1

The PADS-2000 Shell is as follows:

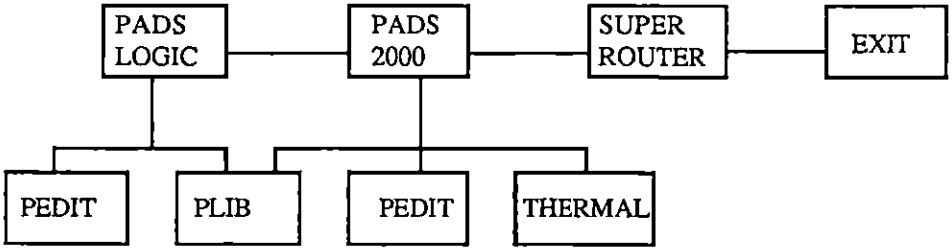


Figure 4.2

To select the SUPERROUTER in either PADS-PCB or PADS-2000, use the mouse cursor and select the Superrouter port.

To enter SUPERROUTER, select the SUPERROUTER Box on the PADS Shell, with the mouse cursor. The CRT presentation shows the Main Menu of Fig. 4.3, and is reproduced as follows:

1 Input
2 Rout
3
4 Strategy
5 Zoom In
6 Zoom Out
7 Save Job
8 Redraw
9 Change Job
0 Exit

Figure 4.3 - MAIN MENU ROUTINES

4.2 LOADING THE BOARD DATABASE FOR AUTOROUTING

The first step, after selecting SUPERROUTER is to Read In the file you wish to autoroute. Read in the file by selecting the INPUT (F1) Menu. If you are reading in a file for the first time, use the function READ JOB (F1). This function takes the PADS database from the \PADS\FILES directory, and converts it into SUPERROUTER format. This newly formatted file has the extension .RUT, and resides in the \PADS\ ROUTER directory.

If the file is an existing SUPERROUTER routed file, select the function READ ROUT (F2). In the instance where the .RUT file does not reside in the \PADS\ROUTER directory, first select WORK DIR (F4), and type in the Path Name of the directory where the .RUT file is located. Normally, you will never use WORK DIR (F4).

Select the READ JOB (F1) option from the menu, and respond with the file name to be routed, omitting the .RUT extension, to the prompt:

Input File Name?

Respond to the prompt:

Delete existing report file (Y/N)?:

If you wish to delete the Report file from the previously routed board, use Y.

Respond to the prompt:

Enter memory allocation for PCB database [1-20 mb]:

Normally respond with the amount of memory calculated as per Chapter 10.

A <CR> response allocates 2 MB.

The software analyzes the available memory in your computer, and the approximate memory required to complete the layout. Memory required is based on the routing cell map, and is a worse-case estimate of the size of the routes array and the pins array. If the free memory is insufficient, a warning message appears. One message that may appear is:

Warning: You have nnnn bytes of free memory. The layout requires xxx bytes. The software will use the hard disk for swapping. Press a key to continue.

Respond to the prompt:

Pause if messages fill screen (Y/N):

A"Y" will pause the CRT message for your analysis.

Respond to the prompt:

Enter Plane Net Names [max 12]: *(PADS-PCB Host only)*

with the names of the nets that are NOT to be routed but rather will be used to create planes or "Exit Vias" in the instance of SMD components.

The prompt will appear with files created in PADS-PCB only. Respond with a Net Name followed by <CR> until all plane nets have been identified. A <CR> will terminate the query.

For files created in PADS-2000, the above query will not appear; instead, while in PADS-2000, use SETUP, NET DISPLAY, identify the Plane Nets and respond YES to PLANES.

In either PADS-PCB or PADS-2000, the above action will result in the following:

1. No routing of these Plane Nets to pins on Thru Hole components will occur. (The plane will be connected to the pin with an inner layer Thermal pad.)

2. An "Exit Via" consisting of a short length of a short track and a via to all pads using Fingers in the pad stack, i.e., SMD parts, Edge Finger Connections, etc. will occur.

Respond to the prompt:

Routing Grid [25]: (*)

with the desired Routing Grid. Legal responses are 10, 12.5, 16.67, 20, 25 and 50 for files created in PADS-2000. Response for PADS-PCB files are 10, 12, 16, 20, 15 and 50.

Respond to the prompt:

Min Clearance [13]: (*)

with the clearance between tracks, pads and vias you wish the Router to observe during the Routing process.

Respond to the prompt:

Default Track Width [12] (*)

with the width of the major track contained in the file.

*NOTE 1: Default Grid, Clearance and Track Widths may differ from the data in the file brought from PADS. Take care to respond as you wish the Router to perform.

NOTE 2: The track width and clearance entered above must NOT exceed the Routing Grid. The Router will not accept a file that does not meet this criteria.

NOTE 3: In the instance of a board with multiple track widths, say 12 mil and 20 mil Signal and 50 mil Power with 13 mil Clearance, and routing on a 25 mil grid. You should NOT enter 50 or 20 mils for Power; instead, respond 12 to the Track Width prompt. During file read in, the 20 and 50 mil wide tracks will be identified as "too wide for Ripup." They will be routed, however, and Optimized.

It will take some time to read the file into memory. You should allow about one second per connection. A series of messages are displayed at the CRT and a number of messages are displayed in the prompt line, as the file is read and analyzed.

Initializing Routing Grid
This will take time as the routing grid is built up.

Writing Connections

Current Net: xxx
Each net name is briefly displayed, as it is processed.

Write Preroutes

The board outline and component pins are displayed at the CRT. Then each connection is drawn as the message:

Conn xxx Width:

is displayed on the prompt line.

Any pre-routed tracks in the layout will then be read into the router and displayed.

During the Loading of Prerouted tracks, a check is made of the Prerouted tracks; and the following action is taken on the preroutes:

- Fixed tracks - No action Taken. Read in "AS IS".

- Tracks too Close to Board Outline - Converted back to connections.

- Tracks whose width is too wide for Ripup possibly - Converted back to connections.

- Tracks with Spacing Violations - Converted back to connections.

- 90 degree corners in unfixed tracks - Will be converted to 45 degree corners, unless blocked. If blocked, will be converted to connections.

When the entire File READ is completed, the message:

READ FILE DONE

appears in the prompt area.

4.3 SET WIDTH (F2)

If you do not want to load Pre-routed Tracks in the Layout:

You may not want to read in the tracks from a Pre-routed Board, but instead start the routing with all the routed tracks converted back to connections. This is particularly true, if you have tried to route the layout once and have made major changes, or want to route on a different grid, with additional layers, etc. If so, you can do this by first Selecting SET WIDTH (F3) option. BEFORE READING IN THE LAYOUT, Select INPUT (F1), then SET WIDTH (F3). To the prompt:

Input New Line Width?

Type a value, in mils, followed by <CR>. This value is displayed in the Router Information Window. Then select the Read option to load the data file. All tracks with a width smaller than this value are converted back to connections during the read process. All tracks with a larger value are kept as tracks. If you wish to Re-route an already routed board, and want to Re-route the entire board, enter some value larger than the already routed tracks, such as 100 <CR>. All tracks

smaller than 100 will be converted back to connections and will be rerouted.

4.4 SELECTING THE DIRECTION RULES FOR LAYER ROUTING

This command, LAYER ASSIGNMENT (F6), is used, only if you have not established the Rules for Layer Direction in PADS. The SR PARMS command is used in PADS-2000, while PAIRED LAYERS is used in PADS-PCB for this purpose.

There is a preferred routing direction for each layer of the layout. Four routing rules are allowed on each layer: (1) Horizontal, (2) Vertical, (3) No Preferred Direction, or (4) No Routing Allowed. The default assignment is alternating Horizontal and Vertical routing on each of the layers in the layout. This may not be desirable for the particular board. For example, some of the layers may be Power planes and should not have any routing, or there may be pre-routes on layers that conflict in direction with the default settings. You may redefine the preferred routing direction by Selecting LAYER ASGN (F6) within the INPUT (F1) menu.

A Pop-up Menu labeled ASSIGN LAYER DIRECTIONS, is displayed in the viewing Window, listing the preferred routing direction for each layer. You may select the layer to be changed with F1; as a layer is selected, it turns red. The Routing Rule for the selected layer is toggled with F2 to one of four allowed settings: Horizontal, Vertical, No Preferred Direction, or No Routing allowed.

When all changes have been made, exit with F10 to update the settings. The Pop-up Menu will remain in the display. The Layer rules will be stored with your .RUT router file, but must be reset, if you create a new .RUT data file.

4.5 SETTING THE ROUTER STRATEGY

The Strategy Control Section allows the Designer to influence routing by setting the costing parameters used to guide the router. You can tell the router the trade-off; for example, between adding an additional via,

or adding an additional route etch. Strategy Parameters may be changed and saved for use on later boards.

The "Standard", or Default Strategy parameter settings have been created for a wide range of boards, and you should take care when deciding to change these values. These values are in a File called DEFAULTS.PAR, and are contained in the PADS Directory. This file is always READ into SUPERROUTER at the time you read in the File to be routed, using the READ (F1) Command. Do not change a parameter, without knowing the results. You may degrade the Router performance, or run time by indiscriminately making Strategy changes.

To view or change the Strategy Table, Select STRATEGY (F4); the menu changes to Fig.4.4 - STRATEGY, as shown in the following:

1 Strategy
2
3 Load Strat
4 Save Strat
0 Exit

FIG. 4.4- STRATEGY MENU

Select STRATEGY (F1) and a Pop-up Menu labeled, VIEW STRATEGY ATTRIBUTES is displayed, listing all of the strategies, their current settings and the allowed range.

You may save the current parameter settings, for use on a later Board layout, by exiting and Selecting SAVE STRAT (F4). When SAVE STRAT (F4) is selected, respond to the prompt:

Input Parameter File Name

Select the name you wish to be associated with the saved Strategy file; a file will be created using the name and extension, .PAR. You may load previously Stored Strategy files from the ROUTER sub-directory with the LOAD STRAT (F3) Menu, and responding to the prompt with the name of the SAVED FILE.

You will notice, when viewing the STRATEGY presentation at the CRT, each of the 18 parameters has a pre-set or default value. These

values are contained in a binary file, in the \PADS\ROUTER Directory called, DEFAULTS.PAR.

Should you find that you prefer a different STRATEGY setup file from the Default, proceed as follows:

1) Call up the VIEW STRATEGY ATTRIBUTES (F1) Window.

2) Make the changes desired using NEXT (F1) and TOGGLE (F2). Then respond to the prompts.

3) Save the changed values into a file using your Selected File Name, using SAVE STRAT (F4).

4) The Strategy File thus created can, thereafter be called up, and re-used on other boards with similar routing requirements.

NOTE: THE DEFAULTS HAVE BEEN DETERMINED AND SET AFTER ROUTING MANY DIFFERENT TYPE BOARDS AND REPRESENT AN "AVERAGE" SETTING. CHANGE STRATEGY ONLY WHEN YOU KNOW EXACTLY THE IMPLICATIONS OF THE CHANGE.

LOAD STRATEGY (F3) is used to call in a Strategy File other than the Default file loaded with the board. It is called in, AFTER you have Read In the Board File to be Routed. If you have created your own Strategy File, respond with its name. If you do not have your own Strategy, the DEFAULTS File, contained in the subdirectory \PADS, is Read In, at the time the file to be Routed is read in, and will be used.

Refer to Chapter 8 for an explanation of each Strategy parameter.

4.6 ROUTING GRID

The selection of the appropriate routing grid as well as track size, air gap clearance and pad size is absolutely essential to routing success. Without the proper values for these items, the router's success will be compromised: Percentage of completion will be low, and run times will be unacceptable. SUPERROUTER is very flexible in letting you select the routing grid independent of the track sizes and clearance rules. This independence is very useful on today's challenging layouts with

multiple pin and track sizes. However, it does require you make the correct choice when selecting these settings.

The "routing grid" is defined as the DISTANCE BETWEEN THE CENTERS OF ADJACENT PARALLEL TRACKS. This is equivalent to the width of the track, plus the minimum space clearance distance. SUPERROUTER offers you the choice of 6 different grids, depending on your manufacturing rules. These are 10 mils, 12.5 mils, 16.67 mils, 20 mils, 25 mils, and 50 mils with PADS-2000 and 10, 12,16, 20, 15 and 50 mils with PADS-PCB.

While Interactive routing in PADS-2000, a Routing grid should be used that is identical in size to the Routing Grid using in SUPERROUTER. If you intend to use 12.5 or 16.67 during SUPERROUTING, be sure to set PADS-2000 to the same grid. Do NOT set the SUPERROUTER to a Grid of 16.67 and PADS to 16. Always use the same grid.

You should always choose the smallest grid that is equal to, or larger than the sum of the signal track width and the minimum clearance you are using. On a board with 12 mil Signal tracks, 50 mil Power and Ground Track Width and 13 mil Clearance, use a 25 mil Grid. Both Signal, VCC and Ground will be routed, however, VCC and Ground will be flagged as "Too wide for Ripup" and may not be ripped up during RIPUP and REROUTE.

The following are Applications of the various grids:

10 MIL GRID:

The 10 mil grid is used for extremely fine line designs when 2 or 4 tracks are desired between 100 mil pitch pads.

12.5 MIL GRID:

The 12.5 mil (12 mil) grid is used for fine line design where typically, 3 tracks between 100 mil pitch between IC pads is desired.

16.67 MIL GRID:

The 16.67 mil (16 mil) grid is typically used where 2 tracks between 100 mil pitch between IC pins, or 1 track between 50 mil pitch SMD pads is desired.

20 MIL GRID:

The 20 mil grid is an alternate to the 16 mil grid where 2 tracks are desired between 100 mil pitch IC pins.

25 MIL GRID:

The most frequently used routing grid used in PCB design is 25 mil. It is used where one track is desired between 100 mil pitch IC pins, or 50 mil pitch SMD design.

50 MIL GRID:

The .050" grid should be used where the sum of track width and space clearance total more than .025", as might occur in Analog Design.

CAUTION: Take care that you have not allowed the Routing Grid and the Parts Placement Grid to become "out of sync" during the activities that were performed in PADS. Read the section Routing Grid Shift (Sectiopn 4.19) carefully and test for the "out of sync" situation, before leaving PADS.

4.7 TRACK, CLEARANCE AND PAD SIZES

NOTE: The Default values for GRID Track Width and Clearances that appear during the reading into PADS-386 SUPERROUTER are Default settings. You may change these values during the read in in of the FILE to be routed.

After having selected the routing grid during File Read In, one selects the minimum clearance and track size.

Your Pad Size and Shape should be set while in PADS--they may not be changed in SUPERROUTER. Again, selection of the correct size pad in track/clearance relation to the routing grid is absolutely essential.

An exercise in grade school mathematics called "doing your numbers," can be easily used to determine the three variables of PAD SIZE, TRACK WIDTH AND AIR GAP CLEARANCE:

Let's say you wanted to determine the values for these three items that will allow one track between IC pads.

We know the pads of an IC are on a 100 mil pitch. For one track to be routed between the pads, the track should be centered on the 50 mil grid (Routing Grid = 25). The space between the center of the pad and the center of the track is occupied by one half the pad diameter plus the air gap (clearance) plus one half the track width.

This is represented by:

1/2 Pad Dia + Air Gap + 1/2 Track Width $\leq$ 50 mils.

The "Standard" track size/clearance for this situation is 15 mils/10 mils, 10/15, 12/13 and 13/12. By "doing the numbers" with these values, we see that the largest possible Pad Diameter (X) can be easily determined.

1/2 (X) + 15 + 1/2 (10) $\leq$ 50 Routing Grid $\therefore$ Pad Diameter = 60 mils
or
1/2 (X) + 13 + 1/2 (12) $\leq$ 50 Routing Grid $\therefore$ Pad Diameter = 60 mils

Always "Do Your Numbers" to determine pad, track and air gap.

4.8 REFERENCE TABLES

Table 4.5 is a "Quick-Look" reference table for your use in rapidly determining the combinations of pad, track and clearance for various situations. Use this reference table to determine the combination of parameters that most closely fit your routability requirements.

Track/Space	Pad Dia	Via Dia	Routing Grid	Placement Grid	Comments
1. 100 MIL PAD PITCH (IC)					
12/13 13/12 10/15 15/10	60	60	25	25	1 track between pads
10/10	50	50	20	20	2 tracks between pads (See Note 1)
8/8	50	50	16.67	16	2 tracks between pads
6/6	50	50	12.5	25	3 tracks between pads
6/4	50	50	10	10	3 tracks between pads (see Note 2)
2/2	50	50	10	10	4 tracks between pads
2. 50 MIL PAD PITCH (SMD)					
12/13	24	20	25	25	No track routed between pads
8/8	24	20	25	25	1 track between pads
10/10	20	20	25	25	1 track between pads
8/8	20	20	16	16	No tracks between pads
5/5	20	20	10	10	2 tracks between pads
5/5	35	35	25	25	1 track between pads
8/8	24	24	25	25	1 track between pads

Table 4.5 - Reference Table

4.10 Use of Copper

Copper areas (NOT COPPER POUR) are polygon pieces of etch on your circuit board, and are created in PADS using the CREATE COPPER Routine. They are distinguished from tracks, in that they are more complex shapes, not simply lines with a width. The copper area may be an arbitrary polygon shape, or a Copper Line, and may be on any layer of the layout, except Layer zero.

Copper shows up as an unfilled polygon when displayed in the PADS-386 SUPERROUTER.

Copper is used to create "Keep Out" areas for tracks and pads under IC's, for example. Isolated and unnamed copper areas created in PADS are used to define "Keep Out" areas.

Named Copper areas, using the same name as a Net on the board will result in the router's attempting to route the Signal Net thru the Copper area, unless the named Copper is remotely removed from where the Net would normally be routed.

A named connection with two identically named copper areas will route to each copper area from a pin, however, a track will not necessarily be placed between the two copper areas.

POURED COPPER (PADS-2000 only) as distinguished from Copper and created using the Copper Pour features of PADS-2000, will also act as a "keep out" feature for routing.

In "Keep Out areas" for either COPPER or COPPER POUR, the Router will not route a track into the keep out areas on the layer containing the copper nor will a via from a track on a layer be permitted to project into the "keep out" area on any other layer.

4.11 TREATMENT OF TEXT AND 2-D LINE ITEMS

Text is treated as if it were a piece of Copper the same size as the Text.

2-D Lines are not passed on to the Router, and hence, are ignored.

Combined Text and 2-D Lines are treated as 2-D lines, i.e., ignored.

4.12 SETTING THE ROUTED LAYERS

PADS-386 SUPERROUTER uses, as the number of routing layers, the number of layers in the File at the time of transfer from PADS. It does not automatically add layers. This is the preferred method of designing PCB's, but it does require you to make a judgement of the number of layers required to route your design while you are in PADS. Set this number in SETUP, SR PARAMETERS in PADS-2000 or SETUP, MISC in PADS-PCB.

Each layer of the board requires substantial memory to run in SUPERROUTER. This is true whether the layer is to be routed, or not. If you do not have sufficient memory based on the calculation inChapter 10 of this appendix, you should set the maximum number of layers in your layout equal to the number of routing layers only. After routing, you can then increase the number of layers to the actual count, including P&G planes, when you return to PADS-2000 or PADS-PCB.

If your system's memory is adequate, you may include the Power and Ground planes and other non-routing layers in the total layer count. These layers MUST be set as non-routing layers in SUPERROUTER, using LAYER ASGN (F6). LAYER ASSIGNMENT permits the setting of each layer to (1) Vertical Routing, (2) Horizontal Routing, (3) No Preferred Direction, and (4) No Routes on This Layer.

Pad Stacks must be set correctly for the file to be read into the SUPERROUTER. Specifically, all parts on a multilayer board, including vias, must have their pad stacks defined as multilayer pad stacks. SMD Parts and Edge Finger connectors and similar parts must have the pad stacks set for the layer on which the pad is to occur.

4.13 ROUTING MIXED WIDTH NETS

PADS-386 SUPERROUTER will autoroute nets containing connections with different widths. An example of this is a Power Net with large .100" busses and smaller .025" connection stubs. In the process, however, the Router will attempt to retain the Width assigned to each connection within the net as much as possible, but may not always do so because of the Length Minimization that occurred as the file is read in. If you must have mixed-width nets with specific widths for each

connection in a net, the net should be interactively pre-routed in PADS, then Fixed.

4.14 DEFINING CONNECTIONS NOT TO BE ROUTED (Fixed Connections)

All connections in the .JOB File will be routed, unless they are flagged as "not to be routed." In PADS-2000 this is accomplished in the NET ATTR command of SETUP by setting the ROUTE parameter for the net to NO; in PADS-PCB, this is accomplished in SETUP, NET DISPLAY and setting the Net to OFF. A second method in PADS-PCB is to list the Net as "Fixed" in the SETUP file. These excluded nets will be listed at the CRT during reading in of the file with the message "Excluded from Routing" and will not be displayed at the CRT with the other connections. They will NOT be counted in the connections count during routing and they will NOT be routed in any of the three routing phases.

4.15 PARTIALLY ROUTED TRACKS

PADS has the unique ability to partially interactively route a connection, i.e. to stop a route, before routing all the way to the connection End Pin; thus, part of the connection is routed and the other part is unrouted. For example, you wish to control two thirds of the track pattern for long connection in a Net named ABLE. In PADS simply interactively route the two thirds of the ABLE track that is critical and end the track with END (F8). END will put in a via, with the routed track before the via and the unrouted connection after the via.

PADS-386 SUPERROUTER supports the partially routed Net and will NOT reroute the routed portion and also will NOT route the unrouted connection portion providing the SETUP, NET ATTRIBUTE for the partially routed net is set to NO ROUTE in PADS-2000 and in PADS-PCB is listed in the SETUP file as "Fixed." Such nets have the message "Fixed" when read into SUPERROUTER.

During all phases of Superrouting, the track part will NOT be affected, however, the connection part will be routed during INITIAL ROUTE and RIPUP and then optimized during OPTIMIZE. Such nets are

considered "partially fixed" and are so identified during File "Read In" into SUPERROUTER.

4.16 FIXED PREROUTED TRACKS

You may choose to preroute connections completely in your layout before starting SUPERROUTER. If you preroute connections, insure the tracks are on the same routing grid you plan to use during Superrouting. Next, "Fix" the net by setting the ROUTE parameter for the net to NO in the NET ATTR command (PADS-2000); in PADS-PCB, use the SETUP file for "Fixing" the Net.

If you do not place these tracks on the same grid as used in SUPERROUTER, the router may turn them back to connections and proceed to reroute them. This is because SUPERROUTER performs a space check on these preroutes during Read In, and if they violate minimum spacing rules, they will be turned back to connections. Make sure that the Routing direction of pre-routes is consistent with the layer selection rules you define for the Autorouter, i.e., if Layer Two in PADS is Vertical, then Layer Two in SUPERROUTER should also be Vertical.

Pre-routed and "fixed" routes will not be moved, or changed, during the Route, Ripup or Optimize phases of SUPERROUTER. Typical examples of fixed tracks would be a Power and Ground, an RF circuit, or critical ECL signals, etc., where layout geometry is important to the function of the design.

Use the PADS Autorouter or Interactive Route or Push and Shove to route Power, Ground and Memory Connections, then fix them. Care should be taken to not specify too many signals as "fixed," because fixed routes act as blocks to SUPERROUTER during Initial and Ripup routing, and thus, will affect completion percentage. On dense boards, pre-routing and fixing more than 10% of the connections on a board may very well prevent 100% routing.

NOTE: To summarize "Fixing":

In PADS-PCB, use the SETUP file and flag the net with either F for an entire net and No Routing will occur, or P for a partially routed net and Routing of the connection will occur.

In PADS-2000, use SETUP, NET ATTRIBUTE and set the Route flag for the Net to NO and No Routing will occur or use NO REROUTE and Routing of the connection only will occur.

When "Fixed" or "Partially Fixed," the net will be thus identified at the CRT during the File Read In.

Fixed Nets will not be routed in INITIAL, RIPUP or OPTIMIZE.

The routed portion of Partially routed nets will not be rerouted in INITIAL, RIPUP or OPTIMIZE; the connection portion will be Routed, Ripped up and Optimized.

4.17 SMD EXIT VIAS

PADS-386 SUPERROUTER provides the capability of automatically exiting from SMD or Edge Finger pads, and using an Exit Via to connect the pad to the inner planes of the board. To use this feature on a multi-layer board with SMD's on either top or bottom, use a track width appropriate for the SMD pad (usually 20 mils wide), and assign NO RIPUP in NET ATTRIBUTE to the VCC and GND nets in PADS-2000; in PADS-PCB during File Read In, respond with the Net Name to the prompt:

Enter plane net name:

The net will be read in to the SUPERROUTER but will not be visible at the CRT but will be partially routed from the SMD pads to an Exit Via. (Chapter 5 provides a detailed procedure for this effort.)

4.18 SETTING UP THE PADS DATABASE

In addition to the controls previously discussed in this chapter, both PADS-2000 and PADS-PCB have SETUP features that permit setting up the database. It is essential that a PADS database be "set up," using these features prior to routing in SUPERROUTER.

The SETUP features for PADS-2000 are:
1. The menu SETUP, SR PARMS (section 4.18.1.1) and

2. The menu SETUP, NET ATTR (section 4.18.1.2).

The SETUP feature for PADS-PCB is the SETUP File (section 4.18.3)

4.18.1.1 SETUP, SR PARAMETERS

This PADS menu routine, as listed below, establishes the values used in the "prompts" presented during the reading of the database into PADS-386 SUPERROUTER.

Routing Grid	XX (Toggle)
Default Track Width	XX
Minimum Track Clearance	XX
Board Edge Clearance	XX
Off-Grid Track Corners [Y/N]	
Via Shift Allowed Y/N (16.67 Grid only)	

Level Direction Flags

1 Horiz	2 Vert	3	4
5	6	7	8
9	10	11	12

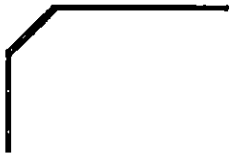
Routing Grid, Track Width and Track Clearance are extracted directly from the PADS-2000 database. You may change these values if you wish as the file is being read into SUPERROUTER. In the process, take care to insure the sum of the Track Width plus the Clearance does not exceed the Routing Grid.

Board Edge Clearance

Establishes the clearance between the inside edge of the Board Outline and the edge of the nearest track to the Board Outline. The Default value is 12 mils. This may be changed.

Off Grid Track Corners

The SUPERROUTER will create 45° corners rather than 90° corners. In the process of creating a 45° corner, thus



the first corner will be "on grid." Depending upon the routing grid selected, the second corner may fall slightly off grid. If you respond Yes to the Off-Grid track query, what are the implications? First, permitting off-grid corners will, in very dense boards, permit 2-3% higher completion than would be the case if off-grid corners were not permitted. Be aware, however, that when the routed board is brought back into PADS, if you expect to do any Interactive modification to the routed tracks, the TRACK MODIFY routine will not "pick up" off-grid corners quite as rapidly as if on grid. On balance, we suggest leaving Corners Off Grid is the preferred approach.

Via Shift Allowed

A response of Yes (only in 16.67 Grid) permits the via center to be moved ± 1/2 the Routing Grid in those instances where the density of the area dictates a Via Shift is necessary to complete the track. We suggest a response of "Y".

Level Direction

Permits the setting of the direction of the majority of the tracks on a particular layer. The control is a "toggle" that toggles between Horizontal, Vertical, Any Direction and None (reserved for Planes). The Default is Horizontal for Odd Numbered Layers and Vertical for Even Numbered Layers.

4.18.1.2 SETUP, NET ATTRIBUTES

This PADS-2000 menu routine is used to select certain "flags" used by the PADS-386 SUPERROUTER and, when selected, presents a selection as follows:

1 Select 2 Add Item 3 Del Item

10 Exit

Signal Name	Disp	LMTYP	Share	Plane	Route	Ripup	Shove
ALL	ON	MIN	YES	NO	YES	YES	YES

A Net (or Nets) can be individualized by selecting ADD ITEM, naming the Net, and setting the flags as appropriate.

DISPLAY OFF

Connections with the SETUP DISPLAY OFF flag will not be displayed at the CRT in PADS-386 SUPERROUTER and will be excluded from Routing. When a File is read into SUPERROUTER, all Nets with DISPLAY OFF will be listed with the message:

Excluded from Routing

LEN MIN

Regardless of the four different minimizations of connections used in PADS during Part Placement, the SUPERROUTER will perform its own Length Minimization as the file is being read into SUPER ROUTER. If you have a net or Nets, where you wish to control the order of the nodes within that net, you should interactively route and fix the net (refer to ROUTE as described below) before reading into SUPERROUTER.

SHARE

NO SHARE on a multi-connection net will result in little or no track sharing between common connections in the Net. Useful for an ECO or High Speed Logic track, otherwise always leave ON.

PLANE

1. DIP Parts

A NO is used to indicate the named Net is not to be routed, and instead will be used to create a Plane. These Nets will not be displayed in SUPERROUTER and will not be routed. However, Surface Mount components will get exit vias automatically for nets listed as a plane net.

2. SMD and Edge Finger Parts

Used to indicate all connections in the net are tied into an "Exit Via" which, in turn, ties into a Plane. The Net will NOT be displayed at the CRT, however, the routing to the "Exit Via" and the "Exit Via" will be routed.

ROUTE

When set to NO for a named Net (Fixing), the net is not displayed in the SUPERROUTER CRT view and is not routed. The CRT screen will display

Fix Net "X"

(where X is the name of the Net with the "No Route" flag) as the file is read into SUPERROUTER. If the net or part of the net or a connection is prerouted, the prerouted track will be displayed in SUPERROUTER but will Not be ripped up and rerouted, and the unrouted connection will not be routed.

RIPUP

Reserved for future use. Do NOT use in current version of SUPERROUTER.

SHOVE

The NET ATTRIBUTE flag of SHOVE is not used by the SUPERROUTER.

4.18.2 THE SETUP FILE FOR PADS-PCB

PADS-PCB uses a Designer created "Setup File" for use in controlling or setting up certain parameters during routing. The file is created in the \PADS\FILES Directory using the PEDIT Text Editor supplied with PADS-PCB. The Setup file name must be the same name of the .JOB File and should use the extension .STP. Example: The Setup file for a job file called PAT would be named PAT.STP.

If, during the reading in of a board file, the message:

No Setup File

appears, this informs you that the Defaults Setup File does not exist for the job. If you indeed have a .STP file, its contents will be displayed (rather than "No Setup File").

The features that can be controlled by the Setup File are as follows:

- Clearance between Board Outline and Tracks.
- The Direction of Routing on each layer.
- Prevent Routing on a Layer (PWR and GND Planes).
- Set all track corners at 45°, or a mixture of 45° and 90° corners.
- Fix previously routed tracks in PADS-PCB, in order that they not be touched by the PADS-386 SUPERROUTER.
- Fix a partially routed track in PADS-PCB in order that the routed portion of the track will not be touched in PADS-386 SUPERROUTER. The connection part will be routed.
- Prevent an unrouted connection net from being routed.

A Default .STP file has been created and is contained in the PADS-386 SUPERROUTER. This Default .STP file has been set as follows:

(Always check to insure this is what you want the Router to do; if not, you must create a .STP file specifically for the job.)

Layer 1 - Routed Horizontal
Layer 2 - Routed Vertical
Corners - All 45° corners (No Off Grid)
No Fixed Tracks - All routed tracks and connections may be routed.
Clearance between Board Outline and Tracks - 12 mils

4.18.2.1 Creating a Setup File

If you desire to change the Setup Default parameters from the above, proceed as follows:

- Select PADS-PCB, PEDIT from the PADS Shell.

- Using PEDIT, create a new file with the same name as the board to be routed, using the extension .STP.

4.18.2.2 Layer Routing Direction

Enter a line of data for each layers, as follows:

First Field, use L for Layer, followed by Space.

Next Field, enter Layer Number, followed by Space.

Next Field, enter H for Horizontal Routing on the Layer or V for Vertical, D for No Preferred Direction, and N for No Routing Permitted. (N is normally used to reserve Layers for Power and Ground planes, if you include these planes in the Number of Layers. Another use is to prevent tracks on the outside layers of SMD boards.)

4.18.2.3 Corner Type

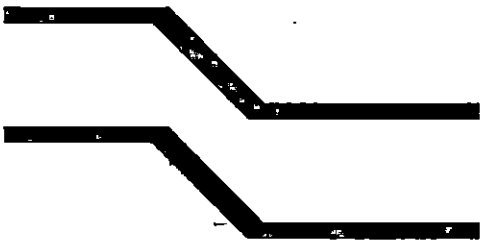
Enter a line of data to command corners on tracks at 45° or 45°/90°.

First Field, enter C for Corners, followed by a space.

Next Field, enter 0 (zero) for both 45° and 90°, or enter 1 for 45° corners only.

You probably want 45° corners on all corners. If so, enter a zero. Be aware that in a few instances, 45° corners can be difficult to modify interactively once you return the database to PADS-PCB. This situation occurs when a corner routed at 45°, falls off grid.

This off-grid corner situation occurs when you have two tracks routed at 45°, and separated by the air gap, as follows:



If indeed, the 45° corners were on grid, the 45° diagonal lines would be separated by slightly less than the required spacing; and thus would violate your spacing criteria. PADS-386 SUPERROUTER detects this potential problem and will space the corners slightly off grid. If you have used a 10 in your Setup File, one corner will be 45°, however, the other Track corner will be 90° (and on grid). If you have used a 1 in the Setup File, all corners will be at 45°, but one will be slightly off grid.

4.18.2.4 Fixed Tracks

Prerouted tracks (the entire net) may be fixed and not touched by the SUPERROUTER.

Enter a data line for each Net that has been previously routed and is to be fixed.

First Field, enter F to indicate the Net is to be fixed and untouched by the Router, followed by a Space.

Next Field, enter the exact Net Signal Name, as contained in the PADS database.

4.18.2.5 Partial Tracks

The feature of "Partial" Tracks covers either one or both of two situations, as follows:

1. The situation where in PADS-PCB, a connection has been partially routed and "ended" before the entire connection has been turned into a track, i.e., part of the connection is a track, and the remainder is still a connection. In this situation, if the net name containing the partially routed connection is flagged with a "P," the routed track portion will be untouched by the SUPERROUTER, and the unrouted connection will be routed.

2. The situation where a net consists of a number of connections, and some of the connections have been routed or turned into tracks, but one or more of the connections has not been routed, and is still a connection. In this situation, a "P" will leave the routed track untouched by the SUPERROUTER, however, the unrouted connections will be routed.

Enter a Data Line for each Net that is partially routed, and is to be fixed as follows:

Enter a P for "Partial," followed by a space.

Next, enter the Net Signal Name, as contained in the PADS database. If you have done extensive Partial prerouting, it is possible to use the wildcard "*" to partially fix all partially routed tracks. Track entry flag would be P * <CR>.

NOTE: Do not use "F *". This would result in no connections being routed by the SUPERROUTER.

4.18.2.6 Fixed Connections

If you do not wish an Unrouted Connection Net to be routed by the PADS-386 SUPERROUTER, treat it as if it were a Fixed Routed Track, i.e., for each connection Net *not to be* routed, create a line of data in the .STP File. The First Field is an F, followed by a space. The Next Field will be the Net Name.

During the Routing phase, the fixed connection will not be displayed at the CRT, and will not be routed by SUPERROUTER.

NOTE: Fixing a track or connection precludes it from being turned Off by the Net List ON/OFF feature in the SETUP Routine in PADS-PCB. That is, a track that has been fixed will be displayed at the CRT in SUPERROUTER, even if turned Off in PADS-PCB Net List Display.

4.18.2.7 Board Outline

If you wish the Router to not route tracks closer to the Board Outline than a specified distance, enter a line of data starting with B (for Board), followed by a space, followed by the desired distance in mils.

Example Setup File:

```
L 1 V
L 2 H
C 1
F ABLE
F BAKER
P CHARLIE
B 50
```

The above Example File would result in the following:

- Layer 1, vertically routed
- Layer 2, horizontally routed
- All 45° corners
- Nets, ABLE and BAKER Fixed and either prerouted or not, will not be disturbed by the Router.

- Prerouted Tracks of Partially Routed Net CHARLIE Fixed and will not be disturbed; the Unrouted connection part of CHARLIE will be routed.
- Tracks will not be routed closer to the Board Outline than 50 mils

To summarize: If you can accept the Default Setup File parameters, you need not take action toward creating a special Setup File. If, on the other hand, you need parameters different from the Default, create your own Setup File for the board to be routed. It will be read in PREAD automatically, when the .JOB File is read in.

During the "Read In," the Signal Name of Fixed and Unfixed tracks/connections will be displayed.

4.19 ROUTING AND PLACEMENT GRID SYNC

The Routing Grid as well as the Placement Grid is established by the System Grid Setting and the Origin location. Moreover, the two Grids must always originate from the same origin point, otherwise the Autorouter will not route as expected, due to the Routing Grid Shift relative to the Placement Grid.

Normally, when placing the parts on a board, a Placement grid is used that is coordinated with the Routing grid. For example, if you expect to route on a grid of 25, you would place the parts using a grid of 25, 50 or 100 rather than a grid of 10, 12.5, 16.67, 20, etc. In such an instance, if you do not change the grid and relocate the origin, there will not be a problem with a shift between the Routing grid and the Placement grid. In fact, even if you relocate the origin but do not change the grid, a problem will not be created.

The problem is created when, after placing the parts with a grid of, say 25, and a small grid, such as G = 1, 2, 5 is selected and, while selected, the origin is relocated, following which, the grid is set back to the original placement routing grid.

What has happened is in the above process the placement grid and the routing grid have been forced out of "synch," thus when a track is routed, it will be off the intended grid by whatever the "out of synch" amount happens to be.

It is important for you to recognize such a situation should you ever inadvertently cause the situation.

Here is how you check for such a situation: Call up your Training board placed, but before Routing has occurred. Make sure the Grid is set to your placement grid (100, 50 or 25).

Using SETUP, place the origin in 3-4 different locations. Notice that the prompt:

New origin is at System Location XXX?? YYY??

is always on the system grid. That is to say, the ?? in both X and Y correspond to multiples of the placement grid (if the Placement Grid is 25, then ?? will always be 25, 50, 75 or 100). This means your Placement Grid and Routing Grid are in synch and the tracks will always be on a 25 mil grid. In fact, to visually see this, Interactively route a track Horizontally between 2 IC pins and zoom up the screen. Notice the track is exactly in the center between the pins, indicating a 25 mil Routing Grid in synch with a 25 mil Placement Grid.

Let's now force the Placement Grid and the Routing Grid "out of sync" and observe the results. Set the Grid to a Grid of 1. Relocate the Origin. Notice the prompt:

New origin is at System Location XXX?? YYY??

no longer has the ?? as multiples of 25. Instead, ?? can be any number between 00 and 25. Example: 36215 16510. The 36215 indicates the routing grid has been shifted 15 mils above the 36200 grid while the Y grid 16110 has been shifted 10 mils above the 16100 grid.

What happens if you were to try to route a track between the 2 IC pins? Remember the parts are placed on a 25 mil grid. Go ahead and try it. Set Grid to 1 and relocate the origin. Notice the ?? is at some value other than a multiple of 25. Try routing the track between pins. Zoom up. Notice the track is no longer exactly between the pins but is offset some distance depending upon the value of ??. If you attempt to autoroute, in all probability, the Autorouter will not route if the "out of sync" is too large.

If you find yourself in such a situation, what do you do to eliminate the "out of sync" condition? The following is an easy procedure that you should use:

1. Set Grid to 1.

2. Place Origin. (Do NOT move the mouse.)

3. What are the values of ??. Let's say they are XXX15 YYY10. You will then want to shift the grid -15 in X and -10 in Y.

4. Set Grid to 25 (or whatever the Routing Grid happens to be).

5. Use the Modeless command of S -15 -10 <CR>

6. Without moving the mouse, reset the Origin. The ?? should now be a multiple of 25 (assuming a Grid of 25).

7. If not successful, repeat until the Out of Sync value (??) indicates the Routing Grid and Placement Grid are again in sync.

4.20 RETURNING A ROUTED FILE TO PADS

The completion of the routing effort is signaled by the prompt:

Save routed connection to job file?

If the pass was practice or you do not want to save the effort, respond:

N <CR>

A **Y <CR>**

response will result in the conversion to PADS format and the prompt:

Update original job file (Y/N)

A Y <CR> will result in the unrouted file being overwritten with the routed file, leaving you without an unrouted file. Normally, respond N <CR>.

Respond to the prompt:

Input new file name

with the name you wish to be associated with the routed file. A file will be created in the PADS/ROUTER directly using the file name with the extension .JOB. This file may be called up in PADS-PCB. A short summary will be given:

Connections Routed	XX
Connections Unrouted	XX
Connections Overflow	XX
Connections Total	XX

together with the prompt:

Delete Router Report (Y/N)

A Y <CR> response will result in the status report of the router being deleted.

Respond to the prompt:

Exit to DOS or Reload SUPERROUTER

as appropriate.

CHAPTER 5

DESIGNING WITH SURFACE MOUNTED DEVICES

PADS-386 SUPERROUTER has a special feature that enhances its usefulness in designing with SMD parts. This special feature is the Automatic creation of an "Exit Via" from the Power and Ground parts to the internal planes.

When using SMD parts, one usually designs with multilayers. Example: 4 layers with Layers 1-4 used for Signal Routing and Layers 2-3 used for VCC/GND Planes, or 6 layers with Layers 1, 2, 5, 6 as Signal Routing layers and 3-4 as VCC/GND Planes. In either instance, the Routing pattern calls for a VCC and GND "Exit Via" from the VCC and GND SMD pads (located on either Layer 1 (top) or Layer 6 (bottom) and connected to the internal VCC/GND planes.

PADS-386 SUPERROUTER incorporates Automatic VCC/GND "Exit Vias," in addition to 100% completion potential for routing the Signal connections.

The general procedure for this effort is to make two passes through the SUPERROUTER: The first pass to route the PWR/GND Exit Vias, then return to PADS to set up for the Signal connections, then make a second pass for the Signal connections.

The remainder of this chapter is a step-by-step procedure for the routing of an SMD 4-layer board with Layers 2/3 as Ground and Power planes. As a suggestion, you may want to use the test board SMDTEST to practice this procedure.

5.1 PREPARATION OF THE DATABASE IN PADS-2000

Perform the following activities on the board database while in PADS-2000:

1. Place the SMD components on the planned Routing Grid. Use 25 mil with SMDTEST.

2. In SETUP, SR PARMS, set all CLEARANCE values.

3. Set to multi-layer as appropriate. (4 layers with Layer 1 Horizontal, Layer 2 Vertical and Layer 2 and 3 No Routes.)

4. In SETUP, SIZE, set Track Widths as appropriate. (Example: Signal 12, PWR/GND at 20 mils or equal to or less than SMD Pad Widths.)

5. Set all Pad Stacks for multilayer. Remember to set STANDARD VIA to multilayer.

6. In NET ATTRIBUTES, set the following conditions:

NAME	DISP	LMTYP	SHARE	PLANE	ROUTE	RIPUP	SHOVE
All	Off	Min	Yes	No	Yes	Yes	___
GND	On	Vert	Yes	Yes	Yes	No	___
VCC	On	Vert	Yes	Yes	Yes	No	___

6. Exit from NET ATTRIBUTE and only PWR/GND will be displayed. Run a LENGTH MINIMIZATION.

7. Save the file.

5.2 TRANSFER TO PADS-386 SUPERROUTER

Using INPUT (F1), read PADS-2000 file into SUPERROUTER. This first routing pass will be for Exit Vias.

1. Read Job.
2. Routing Grid: 25 <CR>
3. Min Clearance: 5 <CR>
4. Track Width: 20 <CR>

NOTE 1: The rationale for using a Grid of 25 and a Min Clearance of 5 is to allow for the PWR/GND width of 20. Had we used a Grid of 25, Clearance of 13 and PWR/GND Width of 20, the sum of PWR/GND (20) plus Clearance (13) would have exceeded the Routing Grid and the file would not have been accepted. The tip off for such a condition is the message provided while reading in the file:

NOTE 2: An indication of proper setup while reading in the file are the following messages:

VCC partially fixed
GND partially fixed
Signals excluded from routing

Suggestion: When reading in the file, respond to the question:

Pause if message fills screen

with a Y. This will allow you time to read and digest the various messages provided.

5. After the file is entered into the SUPERROUTER, no connection will be displayed, however, routing may start. Use ROUT INIT (F2) only. Do NOT, repeat, DO NOT use ROUTE ALL (F1), ROUTE RIPUP (F3) OR ROUT OPT (F4). To do so will destroy the SMD Pad "Exit Vias."

As Initial Routing occurs, observe the Exit Vias that are placed next to each SMD VCC/GND pad.

NOTE: On parts with multiple VCC or GND pads, all connections will be combined into a single "Exit Via."

6. At the end of INIT ROUT, save the file for return to PADS-2000.

5.3 PADS-2000

Return the "Exit Via" routed file to PADS-2000 for:
(1) checking of the TIE PLANES and

(2) setting up the file for routing signal connections.

1. Run CHECK, TIE PLANES and inspect the report. All VCC and GND nodes should have been tied into their plane.

2. Set SETUP, NET ATTRIBUTES as follows for routing the signal connections:

NAME	DISP	LMTYP	SHARE	PLANE	ROUTE	RIPUP	SHOVE
All	On	Min	Yes	No	Yes	Yes	___
GND	Off	___	Yes	Yes	No	No	___
VCC	Off	___	Yes	Yes	No	No	___

3. Run a LENGTH MIN, make a JOB SAVE.

5.4 ROUTING OF SIGNALS IN SUPERROUTER

This phase will route the Signal connections on Layers 1 and 4.

1. Read file saved in PADS-2000 into SUPERROUTER.
2. Routing Grid: 25 <CR>
3. Min Clearance: 8 <CR>
4. Track Width: 8 <CR>

NOTE: During reading of file, the following messages should occur:

GND Width 20 excluded from Routing
VCC Width 20 excluded from Routing
Signals Width 8

The above message indicates the file has been set up correctly for the PWR/GND to not be affected but the signal connections will be routed.

5. Select ROUT ALL for routing.
6. After routing, make a Job Save and transfer back to PADS-2000 for Checking, Plotting.

5.5 PREPARATION OF THE DATABASE IN PADS-PCB

The following procedure is used for creating "Exit Vias" on files prepared in PADS-PCB. Again, use Test File SMDTEST.JOB to become familiar with the procedure.

1. Place the SMD components on the planned Routing Grid, i.e., 10, 12, 16, 20, 25 or 50. (Use 25)

2. In SETUP, set SPACING (8 mils).

3. Set to 4 Layer.

4. Set Signal to 8 mils, PWR/VCC to 20 mils.

5. Set Padstacks to multilayer, including STANDARDVIA (25 mils).

6. In SETUP, NET DISPLAY, set ALL to OFF, VCC and GND to ON.

7. Run Length Min and Save job.

8. Create a SETUP file, using PEDIT with the same name as the saved Job file with VCC and GND set to P (Partial Fixed).

5.6 TRANSFER TO PADS-386 SUPERROUTER

Read into PADS-386 SUPERROUTER for routing of Exit Vias.

1. Read Job.

2. Respond to prompt:

Enter plane net names

with

VCC <CR>
GND <CR>

3. Routing Grid: **25 <CR>**

4. Min Clearance: **5 <CR>**

5. Track Width: **20 <CR>**

NOTE: An indication of proper Setup while reading in the file are the messages:

VCC partially fixed
GND partially fixed
Signals excluded from Routing

After reading in the file, no connection will be displayed, however, routing may start. Use ROUTE INIT only. Do NOT, repeat, DO NOT use ROUTE ALL.

After Routing the Exit Vias, make a Job Save and return to PADS-PCB.

5.7 PREPARATION FOR SIGNAL ROUTING IN PADS-PCB

1. Inspect that all VCC and GND pads are tied to an "Exit Via."

2. In SETUP, DISPLAY, turn ALL to ON and PWR/GND to OFF.

3. Make a Job Save.

4. Create a Setup file using the Job file name assigned in (4) above with VCC and GND using "P" (Partially fixed).

CHAPTER 6

ROUTING WITH PADS-386 SUPERROUTER

After you have set the Layer rules and read in the Strategy Parameters, Select the ROUT (F2) Menu to run the Autorouter. There are four options within the ROUT menu: 1) ROUT ALL, (F1) 2) INITIAL ROUT (F2), 3) ROUT RIPUP (F3), and 4) ROUT OPT (F4). Selecting one of these Router options from the menu will initiate the Autorouter Activity. As SUPERROUTER routes, you will see the results displayed, numerically in the status information window, and graphically in the viewing window. Results of the Autorouter are automatically saved and overwritten into the .RUT file at the end of the INITIAL Router, after EACH track is routed in the RIPUP ROUTER, and after every pass of the OPTIMIZER Router.

6.1 THE ROUT ALL OPTION

Select this option, if you want to run ALL of the Passes in the Autorouter.

NOTE: ALWAYS USE "ROUT ALL" IN YOUR ACTUAL BOARD DESIGN, AND USE THE THREE INDIVIDUAL ROUTING PASSES FOR TEST PURPOSES.

The three sequential passes that are automatically programmed when you select ROUT ALL are as follows:

INITIAL ROUTING (PASS 0):

Four sequential, but different PASSES are as follows:

An INITIAL routing pass, with a maximum non-preferred direction routing of .1", and no vias allowed.

An INITIAL pass with a non-preferred direction routing of .2", and up to 2 vias allowed per connection.

An INITIAL pass with a non-preferred direction routing of .4", and up to 3 vias allowed per connection.

A final INITIAL pass with a no non-preferred direction routing, and as many vias allowed per connection as has been set by the maximum via option in the Strategy parameters.

RIPUP ROUTING:

(Follows the Initial Routing Phase)

A number of Ripup and Reroute Passes, until 100% of the connections are completed. If there is not enough free space to complete 100% Routing, the Ripup pass will continue until terminated.

OPTIMIZER:

(Follows the Ripup Phase)

One or more optimizing passes. This number of passes is defined by STRATEGY Parameter.

6.2 THE INITIAL ROUTER

(This Selection runs the Initial router Only).

The Initial router consists of a series of iterations of the Initial Router, as it attempts to find a solution for all of the connections on the board. (An iteration is an attempt by the Router on every unrouted connection). Those connections that cannot be routed in one iteration are left for succeeding iterations. The Initial Router is organized into several iterations for performance reasons. Severe constraints on the number of vias, routing in the non-preferred direction, and maximum cost of routed paths are placed on the first iteration of the Router, with gradually fewer constraints on those following. Once a path is chosen, it is NOT changed by the Initial Router.

The iterations automatically programmed when you select this option are as follows:

- An initial routing pass, with a maximum channel size of .1", and no vias allowed.
- An initial pass with a maximum channel size of .2", and up to 2 vias allowed per connection.
- An initial pass with a maximum channel size of .4", and up to 3 vias allowed per connection.
- A final initial pass with a no-channel size limit, and as many vias allowed per connection, as has been set by the maximum VIA per connection parameter in the STRATEGY file.

The channel size is the amount of wrong-way routing allowed, e.g. Horizontal routing on a Vertical layer.

The results are written to disk, and the screen is redrawn after each iteration.

6.3 ROUT RIPUP (THE RIPUP ROUTER)

This selection runs the Ripup Router only. The Ripup Router is designed to complete the connections remaining after the Initial Router has finished. The first step taken by the Ripup Router is to examine the initial router pass and the remaining unrouted connections. These connections cannot be completed without ripping up existing routes, so the next step taken by the router is to identify where the Unrouted connections are blocked. The area surrounding each unrouted connection is analyzed to determine which of the existing tracks are blocking a path for the unrouted connection. These are removed (the Ripup), and the Router attempts to find an alternate path for the original Unrouted Connection. Then the Ripped-up tracks are rerouted. This process may require Ripping up many existing routes. To complete the final few of the connections, the Ripup phase may take longer than routing the first 97% of the connections.

If you are an exceptionally skilled and fast PC-Board Designer, you may finish the board sooner if you STOP the Router and optimize the results, and Interactively finish the unroutes yourself, rather than use Ripup and Reroute.

6.4 THE OPTIMIZER ROUTER

(This Selection Activates the Optimizer only).

The optimizer is designed to greatly improve the manufacturability of the board by removing vias, removing bends in the tracks resulting from obstacle hugging routing, and spreading the tracks over the board area. This selection should be run after the Router has completed Ripup, or has achieved 100% completion. In each pass of the Optimizer, the Router tries to improve the quality of every connection on the board by ripping the connections out, and re-routing each connection.

When ROUT OPT (F4) is selected, you are prompted to type the number of optimization passes. When you type a value for the number of passes, the Router starts. This final phase of the Router is designed not to increase routing completion, but to improve manufacturability. (Although in the process of optimization, room may be made for unrouted connections left after Ripup. In this instance, the optimizer will Route the Unrouted connections.) The Router reduces the number of vias and segments, while at the same time improving the quality of the Route paths. The basis for optimization is a Maze Router, with a cost increase for vias and bends in routes. Each route is considered in turn, removed and completely rerouted. It is always possible to complete the route again, because its original path is open; the goal is to improve it. For best results, a number of passes of the optimizer are usually recommended, time permitting (usually 2-3), because as each pass is made, new channels and opportunities are opened for already optimized routes. Beyond a certain number of Optimizing Passes, additional Passes are of little value; specifically, if optimizing at a Speed of SLOW, more than two Passes are of little value. If using a speed of FAST, more than 3-4 Passes does little in the way of improving the board.

The Optimizer Router may be set to either Fast, or Slow optimizing by using the (F8) OPTIMIZER SPEED Menu Software Switch in the F2 ROUTE Menu Command.

The optimized results are written to disk, and the screen is redrawn after each pass of the optimizer.

NOTE: IMPORTANT COMMENT ON ROUTER PASSES

Significantly different (and better) results will be obtained if ROUTE ALL is used during the real board routing effort, rather than the individual routing phases of INITIAL, RIPUP and OPTIMIZE. This is because with ROUT ALL where all three phases are performed sequentially, the memory or intelligence obtained during Initial Routing, is retained and used during RIPUP. Pattern intelligence obtained during RIPUP is retained and used during the OPTIMIZATION Phase. When the three individual routing phases (INITIAL, RIPUP and OPTIMIZATION), are run as separate phases, the pattern memory is not retained and re-used during the next pass. For this reason, use the individual passes for testing; however, use ROUTE ALL for the actual routing effort.

6.5 INFORMATION DISPLAY

As the Router works, the results are displayed in the Router Information Windows (Purple and Yellow). The meaning of this information is as follows:

r pass — The number of the router pass being run. This number is incremented after INITIAL routing is completed, after each unrouted connection is routed in RIPUP, and after each pass of the OPTIMIZER. Each time a pass is completed, the entire design database is automatically stored on the hard disk under the .RUT file; and the display is updated. This insures that you will lose a minimum of routing information, if you have a computer failure.

Conns — Total number of connections in the design file.

Routs — Total number of routes (Connections that have been routed) routed at the current time. This number will decrease slightly, then increase during RIPUP and OPTIMIZATION. Prerouted connections are included in this total.

Vias — Total vias in layout.

% Done — Percentage of total connections completed.

Status	The name of the design currently being autorouted.
Board Size	The X and Y dimensions, in .001", of the layout.
Rout Grid	The selected routing grid, in .001" units.
# Layers	The total number of layers in the layout.
Off Grid	The number of component pins that do NOT fall on the selected Routing grid.

Additionally, other displays giving the input width select by Set Width, the Zoom Factor selected in window, and the speed of the Optimizer you have chosen, appear when these options are activated.

6.6 INTERRUPTING THE SUPERROUTER

The Router may be interrupted at any time by Selecting Key F9 - STOP ROUT. After a brief wait, the software will save all routes completed to that point into the .RUT intermediate file. If the Router is interrupted during the Ripup and Optimizer phases, all ripped out routes will be replaced, automatically.

6.7 SUPERROUTER REPORTS FILE

A report file containing all Router results and user actions is automatically created when you run the Router. The file in ASCII format, is stored in the \PADS\ROUTER Directory, or your Default Input Directory, and takes the name of the Design file with the extension, .RPT. If this file already exists, the new report results are appended to it automatically. Information contained in the report includes routing times and the results for each pass, as well as any options selected by the Router.

To Print out the File, use the DOS PRINT Command. Be in the Subdirectory \PADS\ROUTER and enter PRINT XXX.RPT <CR>.

CHAPTER 7

ROUTING CHECKLIST

This chapter contains a checklist to use while in PADS-2000, laying out a board that will be autorouted in PADS-386 SUPERROUTER. Additionally, a checklist is included for use in setting up the SUPERROUTER after the .JOB FILE has been moved to the SUPERROUTER.

7.1 PADS-2000 CHECKLIST

This Checklist is presented for your use while you are in PADS-2000, and setting up your files for Routing in SUPERROUTER.

1. Establish the placement grid before placing components. Do this in conjunction with the planned routing grid. Eliminate or minimize off-grid pads, or the results may be disappointing.

2. Insure that your pad, track, and via sizes have been selected properly, so that routing channels will not be blocked. Insure that your routing grid is consistent with the rules you plan to use. Calculate and allow for the effect of any off-grid pads.

3. If the power and ground nets are to be on routing layers, route these connections using INTERACTIVE ROUTE, or with the AutoRouter before using the SUPERROUTER to route the signal nets. Fix the PWR & GND Tracks with the NET ATTR, No Route Flag in PADS-2000, or fix in a SETUP file if in PADS-PCB.

4. If there are any partially routed tracks in PADS-2000, partially fix with NET ATTRIBUTE of NO ROUTE, or if in PADS-PCB, use a SETUP file and "P" for Partial Route.

5. Check that the routing direction, selected for each layer is what you plan to use in PADS-386 SUPERROUTER, etc., and any prerouted tracks on the board agree with the same layer and routing direction.

6. If there is an area on the board that you know is so crowded that the connections must be routed in a specific pattern to fit, you should consider pre-routing this area. This comment also applies to Bus Connections. Fix these prerouted tracks.

7. Any pre-routed connections should be on the Routing Grid that you will use in SUPERROUTER. Otherwise, they may be converted back to connections, if a space error exists.

8. Check that there are no spacing errors in the layout by running a Design Rule Check in PADS, and correct any violations. Spacing violations on pre-routes and vias will cause the routes to be converted back to connections.

9. Insure that the connections to be routed are turned ON in DISPLAY.

10. Set the number of layers to the correct number. Check that all pad-stacks, including Standardvia, are also set to the correct number of layers. If multilayer, all pad-stacks, including vias, must be set to multilayer.

11. Insure component pins do not extend off the board, particularly if you have edge connectors. This will cause SUPERROUTER to stop, while it is loading your design.

12. As a file is being read into the PADS-386 SUPERROUTER, an autorouter Length Minimization is performed so as to always have the shortest length to route. If you have a specific sequence of Nodes in your Net that you must retain, or if you want a specific Track pattern after routing, you should route the connections in PADS-2000 and fix them.

13. A signal net with mixed width connections will have these widths observed in the SUPERROUTER insofar as possible. If you do not wish this width, Pre-route in PADS-2000 and Fix in the NET ATTR command by setting the Route attribute to Off or Ripup to Off.

14. Finally, before saving the .JOB file, double check the system grid is set to the planned routing grid, component pads to the correct size, tracks to the planned track width, and clearance to the correct air gap. Check that the Routing Grid and Placement Grid are "in sync" (see Section 4.19).

Check that the Routing Grid and Placement Grid are "in sync" (see Section 4.19).

7.2 PADS-386 SUPERROUTER CHECKLIST

Consider the following Checklist as you are starting the SUPERROUTER:

1. Calculate the amount of memory you need to autoroute the layout to be sure you have adequate memory for SUPERROUTER.

2. Make sure you have sufficient hard disk space to run the Router and store the results.

3. Check all Strategy parameters. Bring in any special Strategy File you may want to use. Otherwise, use the Strategy Default file.

4. CHECK THAT OFF-GRID PADS ARE WITHIN A REASONABLE AMOUNT. If more than 5-10% of pads are off grid, find out why and correct with new placement in PADS-2000.

5. Check the OPTIMIZER SPEED is set to SLOW or FAST, together with the number of Optimizing passes as required.

CHAPTER 8

STRATEGY OPTIONS

This chapter provides detailed information about the meaning of the Strategy Parameter options in SUPERROUTER. For each strategy parameter, the minimum and maximum values are listed, along with the Default Setting. You should change the parameters, only if you understand how they work, and have a specific objective in mind. In general, the parameters have been optimized for typical layouts. Random changes may lead to longer routing times, lower completion percentages, etc. Those parameters flagged as not to be changed should be modified, only at the direction of CAD Software, Inc.

OPTIMIZING PASSES Min [0] Max [99]

Defines the number of optimizing passes to be run during Rout All. The more the number of passes, the better the quality of routing, up to a point, (and of course, longer the time). A minimum of 2 passes is recommended, if you are running the Fast Optimizer; one pass, if you are running the Slow Optimizer. The Default setting is 1. Change to 2, if you use the Slow Optimizer.

OPTIMIZATION RULES Min [0] Max [1]

Defines the rules for manufacturing improvement the Optimizer will employ. Allowed values are 0 for Normal, and 1 for solder wave. Selecting (0) the Normal setting will make the optimizer concentrate on reducing vias. Selecting (1) solder wave will cause the optimizer to reduce routing in the non-preferred direction on each layer, but will increase the via count. Default setting is 0 for Normal Optimization.

VIAS PER CONN Min [0] Max [99]

Defines the maximum number of vias allowed per pin-to-pin connection. You may set the maximum vias the Router will use per connection with this option. If you limit the number of vias allowed, you will increase routing difficulty on large, 2-Layer boards. You might consider reducing the number of vias for multi-layer boards,

where vias block channels on all layers. Default is 20. If Board is a Large Board - larger than 12" X 12", change to 25-30.

VIA COSTS Min [2] Max [20]

Sets the cost for adding a via during Initial and Ripup routing. High values will tend to reduce via count and are recommended, if you are doing boards with many signal layers, because a feedthru uses up real-estate on all layers of the board; and therefore, has a higher relative cost than on 2-layer boards. Default value is 10.

PIN CHANNEL COST Min [0] Max [10]

Sets the cost for using route channels between adjacent pins of IC's. Generally, channels between IC's & SMD in dense digital and SMD designs are blocked because the channels are occupied by connections that traverse thru dense areas. Setting a high pin channel cost will force the routes to go around the dense areas, and restrict the channels to those routes that must use them to be completed, i.e. the connections that start or stop in the dense areas. Value range from 0 to 10; Default value is 3.

NONPREFERRED ROUT COSTS Min [0] Max [5]

Sets the cost of Routing in the non-preferred direction on a layer, e.g., Routing vertically on a preferred horizontal layer. Too many wrong-way segments will block Routing in the preferred direction, and result in poor routing completion. Assigning high value to this parameter tends to create more vias. Value range from 0 to 5; Default value is 2.

NONPREFERRED OPT COSTS Min [0] Max [5]

Sets the cost of routes in the non-preferred direction on a layer during optimization. Assigning a high value to this parameter will tend to create more vias. Value range from 0 to 5; Default value is 2.

ROUT PACKING COST Min [0] Max [5]

"Packing" means placing new routes as close as possible to existing routes, component pins, or obstacles. This tends to leave space for other routes, and is valuable for very dense boards. A high route

packing cost, (3, 4, & 5) will lead to more vias and longer run times. Value range from 0 to 5; Default and suggested value is 1.

DYNAMIC DENSITY COSTS Min [0] Max [50]

As SUPERROUTER works, it dynamically tags the cells adjacent to all routes and vias in the layout. Periodically, it globally analyzes each area of the board; and if many of the cells are tagged, the area is recognized as "dense," and the cost of using a route channel in the area is increased. SUPERROUTER automatically makes it more expensive to find a path for a connection through a densely routed area, than an open area and forces routes that don't have to go through a dense area to avoid it. Value range from 0 to 50; Default value is 10.

RIPUP TRACKS COUNT Min[1] Max [9]

The Ripup Router analyzes the paths blocking non-routes, and then tries to move these paths to other areas in the layout. Unlike other Ripup Routers, SUPERROUTER is capable of removing more than a single blocking track at one time. This parameter sets the maximum number of routes that may be simultaneously removed by the Ripup router to find a path for a non-route. Value range from 1 to 9; Default value is 2.

RIPUP STACK SIZE Min [1] Max [999]

In order to find new Routing channels for routes being ripped up, it may also be necessary to further move additional routes. This is something like a domino effect. The stack size parameter sets the maximum depth of "dominos" that may be re-routed. Value range from 1 to 999; Default value is 50.

RIPUP ITERATIONS Min [0] Max [99]

Sets the maximum number of attempts the Router will make to find a path for each connection, before giving up. Higher values will lead to higher Routing percentage, but it will also take longer. Value range from 0 to 99; Default value is 2.

VIA GRID Min [0] Max [2]

This parameter allows you to limit vias to locations on a fixed grid. A value of 1 limits vias to a .050" grid; a value of 2 imposes a .100" grid. (This .100" grid may be required by your manufacturing, or test departments). Default of 0 places vias on the selected ROUTING grid. Setting this parameter to other than 0 will increase Routing time and could reduce completion results.

BUS COST DURING RIPUP Min [0] Max [5]

Internal parameter used during Ripup. Do not change.

RIPUP DISTANCE 1 Min [0] Max [10]

Internal parameter used during Ripup. Do not change.

RIPUP DISTANCE 2 Min [0] Max [10]

Internal parameter used during Ripup. Do not change.

TRACK CROSSING COST Min [20] Max [100]

Internal parameter used during Ripup. Do not change.

CLEANING DURING RIPUP Min [0] Max [1]

Internal parameter used during Ripup. Do not change.

CHAPTER 9

USING THE PADS-386 SUPERROUTER

In this Chapter, we will discuss:

1. Estimating routing time.

2. Testing the routablility of your board, and

3. Getting the most out of your SUPERROUTER.

9.1 ESTIMATING ROUTING TIME

How long will it take to route your board with SUPERROUTER? That depends on board density, design options and parameters selected, speed of your computer, amount of memory and, of course, the number of connections to be routed. Typical routing time will vary widely. As you will see below, many factors have a complex effect on completion time, depending on the specific design being routed. This Section discusses these factors to give you an idea of the time required to route your boards.

1. Board Dimensions. Larger boards typically will have longer connections requiring more routing time. SUPERROUTER will try to use the entire board area when completing difficult routes, leading to long routing times for some of the connections. A large board of a given density also is, in general, more difficult to route than a small board of the same density.

2. Routing Layers. The number of routing layers impacts routing time in two ways. Increasing the number of layers for a dense board will reduce routing time by increasing possible route paths, and thereby reduces the time required to find a path. The number of layers works to

increase routing time for individual connections by increasing the number of grid cells evaluated during the maze process.

3. Routing Grid. Decreasing the routing grid size will increase routing time. This is because the router must analyze more routing cells per inch of route etch during the sweep process (e.g. 1600 cells per square inch are analyzed with a .025" grid; 10,000 cells, if a .010" grid is used). The number of layers also affects the routing time because the number of cells per inch is multiplied by the number of routing layers.

4. Via grid. Restricting the vias to a 100" grid will increase routing time and decrease Routing completion.

5. Connection Count and Type. The number of connections affects routing: the greater the number of connections, the longer the routing time.

The type of connections, e.g. memory versus random logic, also affects routing time, with memory and short connections taking less time than random, long connections.

6. Density. The greater the component density, the longer the routing time required to find a path for the connection.

7. Computer Hardware. The type of computer you have will have a tremendous effect on performance. AT 386 machines are typically 3-4 times faster than 286 machines. This speed difference increases, if your boards are large enough to exceed the standard 640KB of memory. Within these broad categories of machines, performance will vary based on CPU speed, memory speed, disk speed, the speed of the system Bus, and the presence of Cache memory in the system.

After this discussion, what can you expect? Well, to start with, the run time for SUPERROUTER will be longer than a typical single pass Personal Computer Autorouter. Remember that SUPERROUTER has three major phases: the INITIAL ROUTER, the RIPUP ROUTER, and the OPTIMIZER. Also, SUPERROUTER's goal from the start is to achieve 100% results, so it takes considerably longer analyzing each route it creates to see the effect it will have on later routing. In view of this, do not be surprised if a 150 IC design takes 15 - 20 hours to complete, and a 300 IC board takes in the order of 35 hours. Since you

are routing 24 hours per day, this should not be significant and in any case, is much, much faster than you can manually design the layout.

9.2 TESTING ROUTABILITY OF YOUR BOARD

As you can see from the above, the routing of a large, dense complex board can take a considerable amount of time. And what is more, if you have set up an impossible situation with off-grid pads, pad sizes, etc., it will take an even longer time just to realize there is a problem with the board database. How do you avoid wasting this time?

First, "Do your Numbers" and double check the accuracy as to routing grid, pad size, etc. Are the choices you have made going to give you the result you want? This is especially important if you are expecting to route 2, 3 or even 4 tracks between component pads.

Second, try routing a small test board in the SUPERROUTER, using your assumptions before routing to your large time-consuming board.

For test use in situations where you are not sure if the router will route as you desire (you want two tracks between IC pads, for example); we have included two test files with the SUPERROUTER software. You should use them in testing your assumptions. We strongly recommend that if you not sure of the router's performance, use the small test boards to check your plan. It will take only a few minutes to prove your plan; whereas, routing the real board and finding you are not getting the results you had hoped for will take hours.

The first test board is called ICTEST.JOB. It is a small two layer board with 4 vertical IC's placed side-by-side, on a two-layer board. The IC pads are 60 mil, round, on 100 mil pitch; vias are 60 mils. Tracks are 12 mils wide with a space of 13 mils; VCC and GND are 50 mils. Placement has been on a 100 mil grid. This arrangement will route one track between pads.

If you have an IC board and must achieve two or more tracks between IC pins, change the ICTEST board pads, grid, tracks and clearance to whatever you plan to use in the real board. Try ICTEST, set as your real board in the SUPERROUTER. You will know the answer in only a few minutes.

SMDTEST is a second small, two layer board that has four SMD parts. These pads are on a pitch of 50 mils. The pads are 24 mil. Tracks are 8 mils; space is 8 mils; VCC and GND are 20 mils. The router will route 1 track between pads. Use SMD TEST to test other selections you may have. Should you be using SMD with a pitch different from 50 mils, simply use the ECO routine and replace the 50 mil pitch SMD's with the planned part.

9.3 GETTING THE MOST FROM PADS-2000 SUPERROUTER

Figures 9.1 and 9.2 below depict routing status vs. units of time during various phases of autorouting. It is emphasized that this data is only representative and will vary greatly from board to board.

By comparing the status of routing on your board against that of the figures, you will get a better appreciation of router performance.

Figure 9.1 represents the situation where 100% routing is achieved during initial routing and the "ripup and reroute" phase is never required. Instead, the router advances directly from ROUTE to the OPTIMIZATION phase. This situation is found in medium difficult boards. If you find yourself in the fortunate situation of achieving 100% during the INITIAL Route phase, you can fairly accurately predict the remaining time, depending upon the number of OPTIMIZING passes and optimizer speed. The ratio of total time spent in INITIAL ROUTE compared to OPTIMIZATION will be in the order of 1:1 to approximately 1:3, depending upon the optimization speed and number of optimization passes.

Figure 9.2 represents the situation with the more difficult boards. In this situation, the INITIAL ROUTE pass does NOT achieve 100% routing and the router advances to the Ripup and Reroute phase and upon completion, advances to OPTIMIZATION.

Units of time spent in each phase vary greatly, depending upon the board, and generally fall in the ratio of 1:5:2 to 1:10:3. Nonetheless, by relating the status of your board being routed to Figure 9.2, you will be able to make useful observations. For example, if Initial Routing required four hours, you may assume that Ripup will take in the order of 20-40 hours with Optimize requiring 8-12 hours.

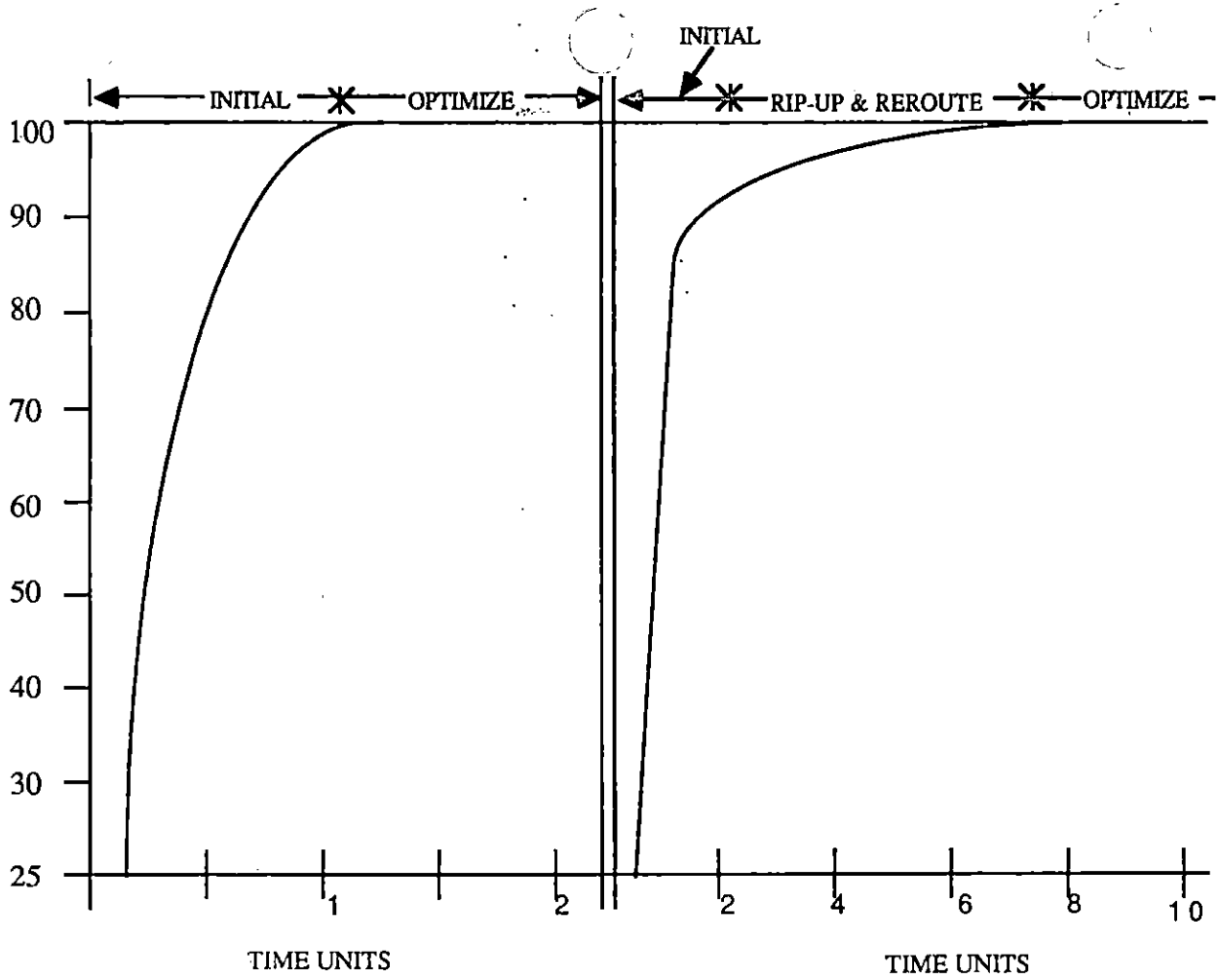


Figure 9.1

Figure 9.2

If you find that the transition from INITIAL ROUTE to RIPUP AND REROUTE has been made with a percentage of completion of about 95% or more, you can be virtually certain the board is 100% routable.

If the percentage of completion at the time of transition between INITIAL ROUTE and RIPUP is below about 92%, you can be virtually certain that 100% routing will not be achievable; you can assume that either the board is too dense or you have, in the process of getting the board ready for routing, created an undesirable situation, and you will need to do some troubleshooting. The following are the most likely candidates:

(1) Have you prerouted and fixed so many tracks that the router does not have the freedom to search? Fixing more than 10-20% of the connections (memory areas excluded) will normally make 100% routing doubtful.

(2) Are the Layer Direction Assignments, in the prerouted fixed tracks the same as those used by the SUPERROUTER?

(3) Have you observed the routing grid size during component placement?

(4) Are the pad sizes, track width and air gap clearance compatible? "Do your Numbers" again. A common error is to set the IC pads correctly but leave the Via at 60 mils.

(5) Look carefully at the CRT. If you expect one track between component pads, are you getting the one track? If you expect 2 tracks, are you getting 2 tracks between pads? If not, the problem is either (3) or (4) above, or both. Recheck. Also check that the Placement Grid and Routing Grid are "in sync" (see Section 4.19).

The major unknown at the end of INITIAL ROUTE is the "time" to complete RIPUP and OPTIMIZE. Let's say you have a board with 4000 connections and the INITIAL ROUTER routed 96%. This means 4% or 160 connections remain to be routed during the Ripup phase. This time can be substantial. Specifically, the Ripup time will be (160 x Strategy Stack Up Depth x Time to Route a connection), plus (Routes not routed in proceeding ripup x stack up x time/route), plus, (etc.),

(etc). If time permits, the proper course to follow is to permit the router to complete Ripup and achieve 100%; then optimize.

If time is of the essence, is there an alternate course? Yes, possibly. If you are an experienced PC Board Designer and are especially fast at interactive routing, it is possible that you can complete the 160 non-routes in a shorter time than Ripup.

Which course would you follow? Our experience suggests that if in the process of interactively routing, you can maintain an average of about 4 minutes per connection, you may complete the non-routes before the PADS-386 SUPERROUTER. If your capability is slower than 4 min./connection, then you should let the SUPERROUTER finish. Remember that the PADS-386 SUPERROUTER is routing 24 hours around the clock, while you are probably on a shift basis.

If you elect to interactively route the remaining connections, we suggest you stop the router at the end of INITIAL ROUTE and then go to OPTIMIZE and complete optimization before returning to PADS. Return to PADS and complete the 160 routes interactively.

CHAPTER 10

MEMORY REQUIREMENTS

PADS-386 SUPERROUTER requires substantially more memory than other autorouters because of its sophistication and flexibility. **You may well need to purchase additional memory for your personal computer in order to run SUPERROUTER.** It is very important that you carefully evaluate the memory needed to route your layouts, and provide sufficient memory in your computer to do the layouts.

The memory available can be considered as a large pool of spare bytes, available to SUPERROUTER and to the system for different purposes. Among these purposes is the operating system, maintaining of the cell map, storing the obstacle data, graphics requirements, program and computational memory space, and memory space for maintaining the connection and routing results. Memory may also be used by other memory-resident programs. It is impossible, therefore, to speak about fixed limits because the allocation of this memory pool is constantly changing, as the program runs. Your memory requirement is the sum of all of the SUPERROUTER memory requirements.

SUPERROUTER requires a *minimum* of 640K of memory. This will be enough, only if your designs are small 50 IC size boards. For larger boards, additional memory is absolutely mandatory! Wherever possible, you should stop any memory resident programs that may be running before beginning the routing.

SUPERROUTER supports normal Extended memory above the 640KB memory limit of DOS. SUPERROUTER has a special protected mode memory handler to access beyond the MS-DOS limit. Up to 16MB of memory may be added, allowing you to do an unlimited size layout. You should calculate the memory required for your boards and plan to purchase extended memory as required for your largest size board.

Where you are designing larger than 200 IC boards, insure that you have at least 16 MB FREE space on your Hard Disk.

You should do a calculation based on your typical board designs using the formula below to determine the amount of memory you will require.

M = A + B + C + D
M = SYS + RA/1024 + PA/1024 + CELL where:
L is the number of layers.

Two examples:

1. The first board is a 4-layer board with 2 signal layers and 2 power/ground planes, 5" x 10", .025" routing grid, and 850 connections and 100 equivalent 14 Pin IC's. Memory required is:

M=350KB+850*160/1024+100*14*30/1024+ (10*40)*(5*40)*4/1024

= 350 KB + 133 KB + 41KB + 313 KB

= 837 KB - Use 1MB of Memory.

2. The second board is a 12" x 15", fine-line, SMT board with 8 layers, 6 signal and 2 power plans, .020" routing grid, 2700 connections, and 250 equivalent IC's.

M = 350KB+ 2700 *160/1024 + 250 *14*30/1024 + (12*50)*(15*50)*8/1024

= 350 KB + 421KB + 102 KB + 3,515 KB

= 4,388 KB = 5 MB

CHAPTER 11

ERRORS AND TROUBLESHOOTING

11.1 ERROR MESSAGES

This chapter lists the error messages from SUPERROUTER, and suggested solutions to the errors. Many of the error messages indicate some form of program error. These should be reported to CAD Software, Inc., along with a copy of the .RUT router file, your original design file, and the report file created by the router.

System does not have 80386 processor. You have loaded the SUPERROUTER onto your PC-AT, or compatible computer. You need to have a 80386 computer.

Invalid Data in Router Workfile. Wrong software revision or database corruption. Call CAD Software, Inc.

Illegal Routing Grid. You have selected a routing grid in your interface that is not supported. Set Grid to either 10, 12.5, 16.67, 20, 25 or 50.

Corrupted Database. Pin ID <>to net ID. Database corruption. Examine Net Identifiers to determine problem IC Pins.

Corrupted Database. Via ID <> to net ID. Database corruption. Examine Via & Net to determine problem.

Error Reading .JOB File. .JOB File, by that name, does not exist in the \PADS\FILES Directory.

Internal Program Error. Database corruption. Call CAD Software, Inc.

Disk Full or Not Enough Memory. You do not have sufficient space on disk to store the results.

XX Too Wide for Ripup File. Net XX track width is wider than the Routing Grid and this will not be ripped up during the Ripup Phase.

Multiple Connections. Database corruption. Run connections check in PADS-2000.

Invalid Number of Conn Pins Database Corruption. Examine Connector Pins in PADS-2000.

Too Many Fixed Tracks. The limit of 500 fixed tracks has been exceeded.

Short in Copper Area. You have two segments of a single copper element overlapping, or two pieces of copper touching. Move copper in PADS-2000.

Short to Pin. Copper and pin specified violate minimum clearance. Move copper in PADS-2000 system.

Via must exist on all routing layers and must be round or square. Via definition error. Change via definition in PADS-2000.

Pin to pin short. Two pins violate minimum clearance. Move one pin.

SUPERROUTER exits to PADS-SHELL. You do not have the blue Security key installed.

Std Line + Clearance > Routing Grid. You have selected incompatible space, track, and grid sizes.

Corrupted Database. Database corruption. Correct Board Outline Error.

Cannot open/write to output file. Disk full or disk error. You need to create space on your disk.

Corrupted Database. Net merge error - Net XX. Database corruption. Examine Net XX.

Internal: Missing Command Name. Database corruption. Call CAD Software, Inc.

Only one file can be loaded. You already have a file in the router. Exit and restart to load new file.

Cannot write parameter file. Either disk full, disk error, or invalid directory specified.

Cannot read parameter file. Either file doesn't exist or invalid directory specified.

Net XXX Excluded From Routed. Net XXX has been turned OFF in the PADS-2000 Net List Display. If you wish the Net to be routed, return to PADS-2000 and turn Net ON.

Error Initializing Graphics. No EGA compatible graphics available.

Copper Area Outside Work Area. You have a piece of copper that extends too far outside of board.

Error No filename Extension Allowed. You may not specify a non-standard filename extension.

Error reading .JOB File - Exiting. System cannot find .JOB File. Check .JOB File Name.

PAD Partially Outside Board Outline. Modify pad on component specified or board outline so that pad is inside board.

Not Enough Memory. You have insufficient memory to load this layout. Add more memory to your system or check for memory resident programs before proceeding.

Program Too Big. Memory Array Connection. Turn CPU off; then Reboot.

Track xx Too Wide for Ripup. Track xx in the database is wider than the width specified during the Read In effort. These tracks will be the first tracks routed in INITIAL ROUTE, but will not be ripped up during the Ripup phase.

Fix Net: Part or all of Net has been prerouted and flagged in NET ATTRIBUTE as NO ROUTE.

Partially Fixed xx. Net has been flagged as NO RIPUP.

11.2 TROUBLESHOOTING

The following is a list of possible problems, and suggested solutions.

Low Routing Completion. This could be caused by many factors. Check the track, space, and pad sizes. Check layer-preferred routing direction rules in PADS-2000. Check the number of off-grid pins in the layout. Review the strategy settings. If you find that an error was made, you should always re-start the routing, rather than continue with the current results.

Results have too many Vias and "Staircases." You need to run further Optimization Passes.

Power and Ground Were Not Routed. You have set these connections to OFF in the PADS-2000 NET ATTR command.

Routing a Connection During Initial Routing Takes Extremely Long. If the time to route a single connection is more than a couple of minutes, check to see if your hard disk drive is constantly active while you are routing. If so, you do not have enough memory; and the router is "paging" to disk-using the disk as temporary memory space. This can be solved only by adding additional memory to your system--a good investment indeed!

Routing a Board With Less than 150 IC's Takes Hours And Hours And Hours! You check the hard disk light, and see that it is always active. The problem is too little RAM Memory. You do not have enough RAM memory, and the system is using its Hard Disk, as a storage Device. You MUST add more RAM Memory.

CHAPTER 12

SUPERROUTER COMMANDS

This chapter provides detailed information about the Menu Commands in PADS-386 SUPERROUTER. For each Menu command, a description of the menu environment, system prompts, user actions, and error conditions are given.

12.1 COMMAND NAME: INPUT

Menu Environment: Main Menu

Function Key: F1

Purpose: To select menu options of the Input Menu

Options:

F1 - Read Job
F2 - Read Rout
F3 - Set Width
F4 - Work Dir
F5 -
F6 - Layer Asgn
F7 - Color Asgn
F8 -
F9 - Read stop
F10 - Exit

Error Conditions: None

12.2 COMMAND NAME: ROUT

Menu Environment: Main Menu

Function Key: F2

Purpose: To select Router Menu Options

Options:

F1 - Rout All
F2 - Rout Init
F3 - Rout Ripup
F4 - Rout Opt
F8 - Opt Speed
F9 - Rout Stop
F10 - Exit

Error Conditions: None

12.3 COMMAND NAME: EXIT

Menu Environment: Main Menu

Function Key: F10

Purpose: To exit from SUPERROUTER back to DOS

System Action:

When selected, the message:

EXIT TO DOS (Y/N)?

appears.

Type: **Y <CR>** to exit back to the PADS-SHELL.

Type: **<CR>, N<CR>, or F10** TO NOT exit.

Error Conditions: None

12.4 COMMAND NAME: STRATEGY

Menu Environment: Main Menu

Function Key: F4

Purpose: To select Strategy Menu Options

Options:

F1 - Strategy
F3 - Load Strat
F4 - Save Strat
F10 - Exit

Error Conditions: None

12.5 COMMAND NAME: REDRAW

Menu Environment: Main Menu

Function Key: F8

Purpose: To redraw viewing window.

Options: None

Error Conditions: None

12.6 COMMAND NAME: READ

Menu Environment: Input Menu

Function Key: F1 (JOB), F2 (ROUT)

Purpose: To select a file for routing from the directory defined in the path.

System action:

The prompt:

INPUT FILE NAME \PATH\NAME

appears. Type in the .RUT or .JOB file you wish to route. The name appears in the prompt line. If the path name is incorrect, you may delete it with the DEL key.

If a routing report file already exists from a previous Routing of the board, the message appears:

DELETE EXISTING REPORT FILE (Y/N):

Respond: Y<CR> or N<CR>

First, the DEFAULTS.PAR Strategy File is Read, then the design file is read, while the prompt line messages are:

INITIALIZING ROUTER GRID
CURRENT NET XXX
WRITE PREROUTES
CONN: nn WIDTH: 0

Any pre-routes are also loaded. When complete, the message is:

FILE READ DONE

You may stop the loading of pre-routes during the Read option by selecting READ STOP (F9).

12.8 COMMAND NAME: WORK DIR

Menu Environment: Input Menu

Function Key: F4

Purpose: To set the pathname of the Work File Directory for reading and writing work files.

System Action:

To the prompt:

NEW WORK FILE DIRECTORY [\ROUTER\]?

Type the name of the path name <CR>. Remember to include any necessary back slash "\" characters before, and after the path name.

This pathname becomes the default path for the input, .RUT design file and the .RPT report file created by the Router.

Error Conditions:

None

NOTE: THIS COMMAND IS FOR USE IN APPLICATIONS OF SUPERROUTER, OTHER THAN WITH PADS. NOT RECOMMENDED FOR USE WITH PADS.

12.9 COMMAND NAME: LAYER ASGN

Menu Environment: Input Menu

Function Key: F6

Purpose: To set the direction of routing for all layers in the layout

System Action:

A pop-up window appears showing the current settings for all routing layers. The bottom layer appears first in the list. Use the function keys F1, F2 to change the settings. F1 (next) allows you to select the next layer in the list for changing. When a layer is selected, it is set Red indicating it is active. After the last layer is selected, the active layer wraps-around to the first layer in the list again.

When the layer you want to change is set red, you use F2 (toggle) to select the desired routing direction for the current active layer. When you Select F2, the routing direction will change to the next setting. Direction options are Vertical, Horizontal, No Preferred Direction, and No routing Allowed.

When all layer settings have been made correctly, Select F10 to exit from the command. The pop-up menu will remain in the Viewing Window. If you want to redraw the viewing area, Select REDRAW (F8) from the top level menu.

The current number of Layers in the pop-up window is determined by the number of layers in PADS.

Layer assignments stay with the file and will be reloaded, if you start the Router again.

12.10 COMMAND NAME: COLOR ASGN

Menu Environment: Input Menu

Function Key: F7

Purpose: To set the display color for all layers.

System Action: COLOR ASGN should be selected, after the database has been loaded into SUPERROUTER.

When selected, a black pop-up menu labeled, Assign Layer Colors appears, displaying the color assignments for each layer. The bottom layer appears first in the list. You use the function keys F1, F2 to change the settings. F1 (next) allows you to select the next layer in the list for changing. The current layer is highlighted in red; you may select a new layer with the F1 key. After the last layer is selected, the command wraps-around to the first again.

The colors can be changed using F2 to toggle the desired color. Colors supported under EGA are red, purple, pink, light grey, dark grey, blue, bright green, light blue, yellow, white, black, dark blue, and green. As you depress F2, the next color appears in the small rectangle to the right of the layer number. When you find the desired color, go to the next layer.

In SUPERROUTER, all items on a layer, including pins, routes, and copper areas, are drawn with the same color. Overlapping items produce a mixed color.

When all color selections are made, Select F10 to Exit. The pop-up menu will remain in the Viewing Window. If you want to redraw the viewing area to see the result of the color changes, Select REDRAW (F8) from the top level menu.

SUGGESTION: FOR YOUR CONVENIENCE, IT IS SUGGESTED THAT YOU USE THE SAME COLOR FOR A LAYER IN SUPERROUTER AS YOU HAVE USED FOR TRACKS ON THAT LAYER WHILE IN PADS. EXAMPLE: IF YOU HAVE YELLOW TRACKS ON LAYER 1 IN PADS, THEN USE YELLOW FOR LAYER 1 IN COLOR ASGN (F6).

Error Conditions:

None

12.11 COMMAND NAME: READ STOP

Menu Environment: Input Menu

Function Key: F9

Purpose: To stop the loading of pre-routed tracks in the Database.

System Action:

Read Stop is selected, while reading a Database into SUPERROUTER. When selected, the message READ STOP appears in the Router Information area. Pre-routed tracks in the design file are not loaded into SUPERROUTER.

Error Conditions:

None

12.12 COMMAND NAME: ROUT ALL

Menu Environment: Rout Menu

Function Key: F1

Purpose: To initiate the Autorouting process with the Initial Router, the Ripup Router, and the Optimizer Router.

System Action:

Select this option, if you want to run all of the passes in the Autorouter. The passes automatically programmed when you select this option are as follows:

Initial Routing: An initial routing pass, with a maximum non-preferred direction routing of .1" and no vias allowed.

An initial pass with a non-preferred direction routing of .2", and up to 2 vias allowed per connection.

An initial pass with a non-preferred direction routing of .4", and up to 3 Vias allowed per connection.

A final initial pass with a no non-preferred direction routing, and as many vias allowed per connection as has been set by the maximum via option in the strategy parameters.

Ripup routing. The Ripup Router continues, until it is unable to find any more route paths.

Optimizer. One or more optimizing passes. This number of passes is defined by one of the strategy parameters.

Error Conditions:

Error messages may occur during routing indicating memory violations, or internal error data corruptions. These errors are listed in Chapter 11.

12.13 COMMAND NAME: ROUT INIT

Menu Environment: Rout Menu

Function Key: F2

Purpose: To initiate the Autorouting process with the Initial Router only.

System Action:

The Initial Router consists of a series of iterations of the Router, as it attempts to find a solution for all of the connections on the board. (An iteration is an attempt by the Router on every unrouted connection). Those connections that cannot be routed in one iteration are left for succeeding iterations. The Initial Router is organized into several iterations for performance reasons; severe constraints on the number of vias, routing in the non-preferred direction, and maximum cost of routed paths are placed on the first iteration of the Router, with gradually fewer constraints on those following. Once a path is chosen, it cannot be removed or changed by the Initial Router.

The iterations automatically programmed when you select this option are as follows:

- An initial routing pass, with a maximum channel size of .1", and no vias allowed.

- An initial pass with a maximum channel size of .2", and up to 2 vias allowed per connection.

- An initial pass with a maximum channel size of .4", and up to 3 vias allowed per connection.

- A final initial pass with a no channel size limit, and as many vias allowed per connection as has been set by the maximum via parameter as set in the strategy parameters. The channel size is the amount of wrong way routing allowed - for example, Horizontal routing on a Vertical layer.

After each iteration, SUPERROUTER automatically saves the result, redraws the graphics, adds the results to the report file, and increments the pass number.

Error Conditions:

Error messages may occur during routing, indicating memory violation, or internal error data corruptions. These errors are listed in Chapter 11.

12.14 COMMAND NAME: ROUT RIPUP

Menu Environment: Rout Menu

Function Key: F3

Purpose: To initiate the autorouting process with the Ripup Router.

System Action:

This selection runs the Ripup Router, and should be run after the initial router has completed. The first step is to run a single iteration of the initial router with no channel limits, and vias limited to the value you have set in the parameter file. The Ripup Router will try to complete every remaining connection in the layout. After each additional track is routed, SUPERROUTER automatically saves the result, redraws the graphics, adds the results to the report file, and increments the pass number.

Error Conditions:

Error messages may occur during routing, indicating memory violations, or internal error data corruptions. These errors are listed in Chapter 11.

12.15 COMMAND NAME: ROUT OPT

Menu Environment: Rout Menu

Function Key: F4

Purpose: To initiate the Autorouting process with the Optimizer Router.

System Action:

When selected, the system prompt comes up:

INPUT NUMBER OF PASS:

You should respond with the number of passes, from 1 to 99, followed by <CR>.

When you type a value with return, the Router starts.

This option runs the Optimizer Router only, and should be run after the Router has completed Ripup, or has achieved 100% completion. The optimizer is designed not to increase routing completion (although it may, if room is made during optimization), but to improve manufacturability. It reduces the number of vias, while at the same time improving the quality of the Route paths. In a pass of the optimizer, each route is considered in turn. It is removed, and completely re-routed. It is always possible to complete the Route again, because its original path is open; the goal is to improve it.

The optimizer may be run in Slow, or Fast mode. These two modes employ different strategies. For best results, several (3-4) passes of the fast optimizer, or 2-3 Passes of the slow optimizer are usually recommended.

Error Conditions:

Error messages may occur during routing, indicating memory violations, or internal error data corruptions. These errors are listed in Chapter 11.

12.16 COMMAND NAME: ROUT STOP

Menu Environment: Rout Menu

Function Key: F9

Purpose: To stop the Autorouter

System Action:

Rout Stop may be selected while running any of the Autorouter options in SUPERROUTER. When selected, the message ROUT STOP appears in the Router Information area. The Router makes a final save of the results to disk. If SUPERROUTER is currently in a ripup or optimize pass, the best results are recovered automatically.

Stopping SUPERROUTER may take several minutes, as the software closes temporary files, reconstructs internal data, and saves the results to disk. When finished, a message appears in the prompt area:

ERROR CONDITIONS:

None.

12.17 COMMAND NAME: ZOOM

Menu Environment: Rout Menu

Function Key: F5 (ZOOM IN), F6 (ZOOM OUT)

Purpose: To change the zoom and pan display of the layout

System Action:

You reposition the cursor over the area on the layout you want to be in the center of the viewing window, and select F5, ZOOM IN to increase the ZOOM factor, or F6, ZOOM OUT to decrease the ZOOM factor. The graphics is redrawn at the new location.

Error Conditions:

None

ADDITIONAL WINDOW CONTROLS:

To PAN, use the Right, Left, Up, Down Arrow Keys of the Numeric Key Board.

To Reset and Recenter the board at the CRT, use Numeric Key 7 (Home).

NOTE: CRT controls are not active during routing activities.

12.18 COMMAND NAME: OPT SPEED

Menu Environment: Rout Menu

Function Key: F8

Purpose: To toggle between the fast and slow modes of Router Optimizer

System Action:

One of the strategy parameters is the Optimizer speed, which may be set either slow, or fast. Because this parameter is one that users will select repeatedly, it is selectable as a menu function, as well as a Strategy Parameter Setting. The Function F8 OPT SPEED operates as a toggle between the Fast optimizer option, and the Slow option.

You should select the Fast option for optimizing when your computer time is limited. The fast optimizer will ripup each net and re-route it to reduce stair steps and vias. The fast optimizer will evaluate all nets in a reasonably short time, and the results will be a considerably better layout. A minimum of two passes of the fast optimizer are recommended (3-4 if time permits), (or use SLOW).

You should select the slow optimizer when you have substantial computer time, or when in PADS-PCB, the number of segments after routing a large board exceeds the PADS-PCB segment limit and you want to produce the best possible routing result. The slow optimizer evaluates each net exhaustively, swapping the tracks with all neighboring tracks to get the best results. You need only a single (or 2 at most), Slow Optimizing PASS.

The Default setting for Opt Speed is fast. When selected, the new setting is displayed in the Router Information Area.

Error Conditions:

None

12.19 COMMAND NAME: STRATEGY

Menu Environment: Strategy Menu

Function Key: F1

Purpose: To make changes to the Router Strategies

System Action:

When selected, the Pop-up Menu labeled, "View Strategy Attributes," appears in the Viewing Window. This lists all of the costing parameters in the SUPERROUTER, and gives the current value, as well as the range of allowed values.

You may select the strategy you wish to change by repeated use of the F1 key. When the key is selected, the next strategy is highlighted in Red and becomes the current strategy. After the last attribute is selected, the Command wraps-around to the first again.

To change the current setting, select the F2 key. The message:

Input New Value

appears, and you should type in a value within the min and max range, followed by <CR>.

Exit by selecting F10, when all settings have been completed.

Error Conditions:

None

12.20 COMMAND NAME: LOAD STRAT

Menu Environment: Strategy Menu

Function Key: F3

Purpose: To read in a previously saved set of Router strategies from a disk file.

System Action:

When selected, you get the prompt:

INPUT PARAMETER FILE NAME:

You are asked type the name of the parameter file previously saved. The Default Directory may be changed, as well. Type the name, followed by <CR>. The parameter file is stored in the current Default Directory.

Error Conditions:

1. Typing a non-existent parameter file gives the message:

CAN'T READ PARAMETER FILE

NOTE: ALWAYS LOAD YOUR OWN STRATEGY FILE AFTER YOUR DESIGN HAS BEEN LOADED INTO SUPERROUTER, IF YOU DO NOT WANT TO USE THE DEFAULTS STRATEGY FILE CONTAINED IN THE SOFTWARE.

12.21 COMMAND NAME: SAVE STRAT

Menu Environment: Strategy Menu

Function Key: F4

Purpose: To store the current strategy settings to disk for later use.

System Action:

When selected, you get the prompt:

Input Parameter File Name

You are asked to type the name of the parameter file you wish to save. Simply, make the change and save as a file Name of your choice. Type the name, followed by <CR>. The changed parameter settings file will be placed in the \PADS\ROUTER Directory.

Error Conditions:

1. Saving the file to a Non-existent Directory. Having insufficient space on the disk and specifying an extension causes the message:

CAN'T WRITE PARAMETER FILE

INDEX

A

ASSIGN LAYER DIRECTIONS 17

B

Board Edge Clearance 31
Board Outline 39
BUS COST DURING RIPUP 64

C

Checklist 2, 57
CLEANING DURING RIPUP 64
COLOR ASGN 90
CONTROLS 11
Controls for PADS-386 SUPERROUTER 2
Corner Type 37
Critical tracks 10

D

Default Track Width 6, 14
DEFAULTS.PAR 18
DEFINING CONNECTIONS NOT TO BE 28
DIP Parts 34
Direction of Routing 10
DIRECTION RULES 17
DISPLAY OFF 33
DYNAMIC DENSITY COSTS 63

E

Error Messages 2, 75
ESTIMATING ROUTING TIME 65
EXIT 82
Exit Vias 10

F

Fast 54
FEATURES 9
FIXED 10
Fixed Connections 39
FIXED PREROUTED TRACKS 29
Fixed tracks 16, 37

I

INITIAL 52
INITIAL ROUTE 9
Initial router 52
Initializing Routing Grid 15
INPUT 80
INPUT MENU 4
Input New Line Width? 16
INSTALLATION TESTING 3
INTERRUPTING THE SUPERROUTER 56

L

LAYER ASGN 89
LAYER ASSIGNMENT 17
Layer Routing Direction 36
LEN MIN 33
Level Direction 32
LOAD STRAT 102
LOAD STRATEGY 19
LOADING THE BOARD DATABASE 12

M

MAIN MENU 4, 7, 12
Memory allocation 13
Memory Requirements 2, 73
Min Clearance 6, 14

N

NONPREFERRED OPT COSTS 62

NONPREFERRED ROUT COSTS 62

O

OFF GRID PADS 24
Off Grid Track Corners 32
OPT SPEED 100
OPTIMIZATION 9
OPTIMIZATION RULES 61
OPTIMIZER 52
OPTIMIZER ROUTER 54
OPTIMIZER SPEED 54
OPTIMIZING PASSES 61

P

PADS Shell 8
Partial Tracks 38
PARTIALLY ROUTED TRACKS 28
PIN CHANNEL COST 62
PLANE 34

Q

"Quick-look" Summary 2

R

READ 85
READ STOP 92
REDRAW 84
RIPUP DISTANCE 1, 2, 64
RIPUP ITERATIONS 63
RIPUP STACK SIZE 63
RIPUP TRACKS COUNT 63
RIPUP 34
RIPUP AND RE-ROUTE 9
RIPUP ROUTING 52
ROUT 81
ROUT ALL 51, 93
ROUT INIT 94
ROUT OPT 54, 97

ROUT PACKING COST 62
ROUT RIPUP 96
ROUT RIPUP (THE RIPUP ROUTER) 53
ROUT STOP 98
ROUTE 34
ROUTER STRATEGY 17
ROUTING AND PLACEMENT GRID SYNC 40
ROUTING CHECKLIST 57
Routing Grid 14, 19, 31
Routing grids 1
Routing Grids are 10, 12.5, 16.67, 20, 25 and 50 mils 10
ROUTING MIXED WIDTH NETS 27
RUTTEST 3

S

SAVE STRAT 19, 103
SET WIDTH 87
SETTING THE ROUTED LAYERS 27
SETUP FILE FOR PADS-PCB 35
SETUP, NET ATTRIBUTES 33
SHARE 34
SHOVE 35
Slow 54
SMD and Edge Finger Parts 34
SMD EXIT VIAS 30
Strategy 2, 10, 18, 83, 101
STRATEGY ATTRIBUTES 19
STRATEGY OPTIONS 61
SUPERROUTER REPORTS FILE 56
SURFACE MOUNTED DEVICES 45

T

TESTING ROUTABILITY 67
Testing the Installation 2
TEXT AND 2-D LINE ITEMS 26
Track Clearance 31
TRACK CROSSING COST 64
Track Width 31
TRACK, CLEARANCE AND PAD SIZES 21

TROUBLESHOOTING 78

U

Use of Copper 26

V

VIA COSTS 62
VIA GRID 64
Via Shift Allowed 32
VIAS PER CONN 61

W

WORK DIR 88
Write Preroutes 15
Writing Connections 15

Z

ZOOM 99

APPENDIX A2

BACKGROUND INFORMATION ON PHOTOPLOTTING

MAKING PLOTS ON A PHOTOPLOTTER:

This discussion on Photoplotting is provided to those CAD users who, for the first time, send files to a photoplotting facility for plotting and generally film for artwork.

NOTE: Parameters for the photoplotter you intend to use, and set up during the PINSTALL- Hardware.

Description of Photoplotter Hardware:

Most photoplotters consist of two major hardware units: the plotter unit and the controller. The controller is the unit that reads the plotter commands either from magnetic tape, diskette, or directly on-line through an RS-232 link. The controller decodes the plotter commands and sends the appropriate signals to the plotter unit to generate the plot. The plotter unit consists of a plotting surface upon which the film is laid and a plotting head that contains the light source, a shutter, and a set of apertures through which the light is projected on the film.

The Basic Photoplotter Commands:

There are four basic commands that all photoplotters will accept:

1) Aperture Select: Most photoplotters provide a choice of several apertures of different sizes and shapes. For example: a round aperture with diameter of 50 mils can be used to "flash" a 50 mil pad.

2) Flash Command: This command opens the shutter for a brief moment which causes the light to flash briefly upon the film. If a 50 mil round aperture is currently selected then a 50 mil pad will be exposed on the film.

3) Move Command: This command moves the plotting surface with the shutter closed so that the head is positioned at the co-ordinates specified. Film is NOT exposed.

4)Draw Command: This causes the plotting surface to be moved to the specified co-ordinates with the shutter open. If a 50 mil round aperture filtered for a "Draw" command is currently selected then a 50 mil wide line will be exposed on the film.

The Aperture Wheel:

Every photoplotter has a set of apertures each of which can be selected by the Aperture Select command. On the Gerber photoplotter, there are 24 apertures selectable on an "Aperture Wheel". The aperture select command causes the wheel to rotate so that the correct aperture appears under the light source. Other photoplotters may use some other mechanism to select an aperture but we will use the term "Aperture Wheel" from now on. Also, other photoplotters such as Laser units, may have more than 24 apertures.

Even though the Gerber has only 24 aperture positions, the actual apertures are inter-changeable. Usually, there are many more different aperture sizes and shapes available than there are positions on the wheel. It is not possible to re-load new apertures in the wheel in the middle of a plot, hence only 24 different apertures can be used on a single plot. The number of different line widths used in the plot and the number of different pad size and shape combinations determine the number of different apertures required; you design. Even though different apertures can be loaded on to the wheel between plots, it is a tedious operation for the operator which takes time and limits productivity. Hence, you should attempt to try to pick a set of 24 apertures that can be re-used for several plots in succession. Also, many photoplotter installations have a "Standard" aperture wheel set-up with the most commonly used apertures loaded. If possible, you should try to use only those sizes provided; it does not compromise your design.

The Aperture Table:

In order for the CAM Photoplotter software to generate the correct Aperture Select commands, it must know which aperture shapes and sizes are to be loaded in each of the aperture positions on the aperture wheel. The Aperture Table can be set-up or modified in the CAM

Defaults routine. In order to set-up the aperture table correctly, you should find out the list of different apertures available at the photoplotter installation you intend to use for your designs. If there is a "Standard Aperture Wheel", then enter the apertures used and only make modifications when special design considerations require you to use non-standard apertures. If there is no standard, then pick the most useful set of 24 apertures so that most of your plots can be generated without changing the aperture table.

The CAM software allows any sized aperture from 1 to 999 mils, but it allows only 9 different shapes:

LINE	A round shaped aperture used for line drawing.
SQRL	A square aperture used for drawing rectangular lines.
RND	A solid round aperture used for flashing pads.
SQR	A square aperture used for flashing pads.
OVAL	An oval shaped aperture for flashing oval pads.
RECT	A rectangular aperture for flashing rectangular pads.
ANNL	An annular aperture for flashing "donut" pads.
THER	A "thermal relief" aperture used for Power & Ground planes.
ODD	Any other shaped aperture used for flashing pads.

NOTE: While the PADS-CAM Routine permits aperture sizes up to 999 mils, the photoplotter you use may be less than 999 mils. Always check the maximum size available if using an aperture larger than 250 mils.

It is important to note that there are two special shapes reserved for line drawing (LINE & SQRL). In actual fact, a RND or SQR aperture used for flashing pads can also be used for drawing lines, but line drawing requires different filters to be placed over the aperture than for flashing hence, two different apertures in two different positions must be used even if the pad diameter used is the same as the line thickness. Said another way because of filtering requirement between flash and draw, aperture filtering for lines can also be used for flash of pads, but not visa versa.

It may seem redundant to provide special shaped apertures for oval and rectangular pad flashing, since these pads can be created by lines drawn with either a LINE or SQRL aperture. In fact, pad flashing is considerably faster than line drawing and hence, the OVAL and RECT flash shapes are very useful. If the correct sized OVAL or RECT

aperture is not available, the CAM software will automatically use the appropriate sized LINE or SQRL aperture and draw the pads instead.

Gerber Command Format:

It is useful to know something about the format of the Gerber output files generated by the CAM software so that you can be sure that you have installed your Gerber software correctly and that you are generating the correct output format for the photoplotter you intend to use.

There are a number of different types of controllers used on Gerber photoplotters plus each controller may be set-up differently at different installations. For example, one site may choose to accept metric units and absolute co-ordinates and another may choose English units and incremental co-ordinates. You need to know the characteristics of the photoplotter where you are sending your files, at the time of PADS-PINSTALL.

The Aperture Select Command:

This command is often referred to as a "D" command because it starts with the letter "D".

The format of the command is: D<aperture D-code>*
The aperture D-code is not the aperture position number from 1-24 as one might expect, but a special code number which can be obtained from the following table:

Aperture Position	Aperture D-Code
1	10
2	11
3	12
4	13
5	14
6	15
7	16
8	17
9	18
10	19
11	70

12	71
13	20
14	21
15	22
16	23
17	24
18	25
19	26
20	27
21	28
22	29
23	72
24	73

For example, if a 50 mil round pad aperture were loaded in position 19, then the aperture select command would be: D26*

NOTE: The "*" at the end of the command is the command separating character which appears at the end of all commands.

Flash Command:

This command causes the table to be moved to the specified position and the shutter opened and closed quickly to expose a flash of light through the aperture.

The format of the command is:

X<x co-ordinate> Y<y co-ordinate>D03*

For example the command:

X1000Y1000D03*

would cause the table to move to co-ordinate position (1,000,1000) and flash light through the currently selected aperture. Note: co-ordinate values can be interpreted differently on different controllers as explained later.

Move and Draw Commands:

Move and draw commands are very similar to flash commands. The format for a move command is:

X<x co-ordinate>Y<y co-ordinate>D02*

The format for a draw command is:

X<x co-ordinate>Y<y co-ordinate>D01*

The D02* command causes the shutter to close and the D01* command causes the shutter to open. The D02* and D01* commands are "modal", that is to say they remain in effect until changed. Hence if a command appears without the D02* or D01* it is assumed that the shutter stays in the same state (either open or closed) as the previous command. For example, to draw a square of 1000 units on a side with lower left corner at (1000,1000) the commands would be as follows:

X1000Y1000D02*
X2000Y1000D01*
X2000Y2000*
X1000Y2000*
X1000Y1000*

End of Plot Command:

The last command in the output file which signifies the end of the plot is:

M02*

Co-ordinate Value Formats:

Co-ordinate values aren't always what they appear. Different controllers are set-up differently. There are a number of different options that can affect the way in which co-ordinate values are output:

Units: The units may be either English or Metric. English units are expressed in some fraction of an inch - usually 0.001 inch. Metric units are expressed in some fraction of a millimeter - usually 0.01 mm.

Precision: This is a number that specifies the number of significant digits in the fractional part of the co-ordinate values. A value of 2 indicates coordinate values in 0.01 mm for metric or 0.01 in. for English units. A value of 4 for English units indicates co-ordinate values of 0.0001 in.

No of Digits: This specifies the maximum number of digits that can appear in a co-ordinate value. If the Precision = 4 and the no. of digits = 8, then the maximum co-ordinate value can be 9999.9999 inches.

NOTE: Co-ordinate values always appear without the decimal point, so the maximum value would appear in the output as 99999999.

Absolute: Absolute co-ordinates means that co-ordinates are expressed as their actual values with respect to the plotter origin.

Incremental: Incremental co-ordinates means that values are expressed as differences from the previous command.

Suppress Zeros:Either leading zeros, trailing zeros, or no zeros can be suppressed from co-ordinate values.

Repeat Values: If either the X or Y co-ordinate values are the same from one command to the next, then the X or Y value may be omitted from the command.

The effect of these options is demonstrated in the examples below. All the examples will be the commands to draw a one inch square starting one inch from the origin in the X & Y directions. All the examples will have precision = 3 and No. of digits = 8.

```
X1000Y1000D02*
X2000Y1000D01*
X2000Y2000*
X1000Y2000*
X1000Y1000*
```

Incremental co-ordinates with leading zeros suppressed:

```
X1000Y1000D02*
X1000Y0D01*
X0Y1000*
```

```
X-1000Y0*
X0Y-1000*
```

Absolute co-ordinates with no zeros suppressed:

```
X00001000Y00001000D02*
X00002000Y00001000D01*
X00002000Y00002000*
X00001000Y00002000*
X00001000Y00001000*
```

Absolute co-ordinates with trailing zeros suppressed:

```
X00001Y00001D02*
X00002Y00001D01*
X00002Y00002*
X00001Y00002*
X00001Y00001*
```

Absolute co-ordinates with leading zeros and repeat values suppressed:

```
X1000Y1000D02*
X2000D01*
Y2000*
X1000*
Y1000*
```

Incremental co-ordinates with leading zeros and repeat values suppressed:

```
X1000Y1000D02*
X1000D01*
Y1000*
X-1000*
Y1000*
```

NOTE: Suppressing zeros (either leading or trailing) and suppressing repeat values makes the output file much shorter.

Gerber Software Installation Parameters:

Most of the installation parameters are to specify the co-ordinate value format options as explained above. You must find out what options are used for the photoplotter you intend to use, otherwise your plots will not work. In addition to the format options, you should find out the size of the plotting area on the photoplotter and enter the size limits in units of 0.001 in. (mils), regardless of the actual units and precision selected, when running the PADS-2000 installation program.

Check-List for Generating Correct Gerber Plot Files:

The first time that you make Gerber plots, you should run through this entire check-list. Also later, if you encounter any difficulties getting your plot files plotted, you should go through this check-list to ensure that everything is set-up correctly.

(1) Check your Gerber Software Installation Parameters. If you are unsure about some of the parameters, consult with the people who run the photoplotter and who know about the setting of the controller format options.

(2) Use the CAM Defaults routine to set-up the "Master" Aperture Table in DEFAULTL.PAD. This aperture table should match the "Standard" aperture wheel for the photoplotter you are using.

(3) Check the pad stack values and line widths that you are using in your design to make sure they are set correctly.

(4) If you are using pad sizes or line widths that are not included in the Master Aperture Table then change as many as you can to match the Master Aperture Table. If you change any sizes, you should re-run space checking to make sure you did not create any problems.

(5) If there are still some pad sizes or line widths that you could not change that still do not match, then you can create an Aperture Table just for the job by changing some of the unused apertures in the Master Aperture Table to match the values used in your job. This is done in the CAM Defaults routine the same way as setting up the Master table except that you do not select the option to update the Master table.

(6) Now you can run the CAM routine to generate your plot files. If while making a plot file you get the message:

Enter Aperture No. for Width: nn, ... etc.

then you still have a mis-match between the job and the aperture table. You are allowed to make temporary corrections "on-the-fly" but this practice is strongly discouraged. You may be better off cancelling the plot and going back to fix up your aperture table or pad stack values.

(7) After making your plot files, you can "verify" your files by plotting them on your matrix printer or pen-plotter using the CAM Verify Routine. The Verify routine is not fool-proof but it will show up the more obvious sizing and spacing errors.

APPENDIX A3

ECO FORWARD/BACKWARD ANNOTATION FORMAT

PURPOSE:

The purpose of this Appendix is to outline the ASCII format of the file that is created by (1) the ECO TO SCHEM routines of PADS-2000 and (2) is read by the ECO FROM SCHEM routine.

The PADS-2000 ECO FROM SCHEM routine includes a subroutine, which will accept as input an ASCII file using the same format as specified herein. This file can be used to automatically update a PADS-2000 design with changes from a schematic capture program, commonly referred to as forward-annotation.

The TO SCHEM file can be used to upgrade a schematic capture program with changes made to the PADS-2000 design, commonly referred to as back-annotation.

TO SCHEM:

There are two groups of routines that can create electrical modifications to the PADS-2000 design; the ECO menu routines, and the GATE/PIN swap and Component Rename routines in PLACE/AUTO, and which, in turn, generate a file to update the schematic (TO SCHEM).

When the ECO menu is selected from PADS-2000 the user is asked to select a file name to which electrical modifications to the design in memory, are written. If the file named already exists, then the option of appending this file or selecting a new file is given.

The GATE/PIN swap routine operates in a manner similar to the ECO menu routine; all swapped pins and gates are reported to a TO SCHEM file.

The COMPONENT RENAME routine operates in a similar manner to the ECO menu; all Renamed components are reported to a TO SCHEM file.

FROM SCHEM:

The ECO menu has a subroutine called FROM SCHEM. When selected, this routine will accept a ASCII file with a list of modifications to update on the PADS-2000 database. Any errors in the file format or inconsistencies in the data are output to a file or the printer.

FORWARD/BACKWARD ANNOTATION FILE FORMAT:

The format used here is similar to the PADS-2000 ASCII input format. Each type of data modified begins with a line of header information. In a header line the key information is located between asterisks (*).

In order to ensure that the file is the correct type of file, a starting header line is used:

PADS-ECO

Once this line is found in an annotation file, the following modification information is processed.

The end of the file (EOF) information header is:

END

An annotation file may have one or even more than one entry from the ECO Routine, or more then one section of modification information in it, so, the file must always be read until the DOS EOF (end of file) marker is reached.

Remarks lines can be added by using the header line:

REMARK remark information etc.

MODIFICATION DATA TYPES:

There are eleven modification data types:

1. Add a pin to net.
2. Join two nets together.
3. Delete a pin from net.
4. Split a net into two nets.
5. Rename a net.
6. Add a part.
7. Delete a part.
8. Change a part's part type.
9. Rename a part.
10. Swap two gates.
11. Swap two pins.

1.Add pin to net.

The header line is:

NET

Always followed by the header line for the net to add the pin to:

SIGNAL NEWNET 10

where NEWNET is the net to add the pins to and 10 is the track width to associate with the connections.

The format for the pin information is:

Y4.3 U7.3 U6.1 U6.8
U9.1
SIGNAL GND 50

To add new nodes (pins) to a new net, simply enter a new signal name header.

U4.7 U6.8 U8.7 U7.8
U9.7

If the net name does not currently exist in the design it will be added.

2. Join two nets together.

The header line is:

JOINNET

The format for the join information is:

OLDNET0 OLDNET1

where OLDNET0 and OLDNET1 are the names of the nets to combine. The new combined net uses the name OLDNET1.

A connection is added between the nets using two random pins in the selected nets. The track width of the added connection is the same as that of a connection in the first net (OLDNET0).

3. Delete pin from net.

The header line is:

DELPIN DELETE CONNECTIONS TO PIN

U17.1 OLK

where U17 is the part reference name, 1 is the pin number to disconnect and OLK is the net that the pin is currently part of (the net name information is not required).

4. Split net into two new nets.

The header line is:

SPLITNET SPLIT NET INTO TWO NEW NETS

The format for the split net information is:

SIGNA C

where C is the old net name of the combined net and the new name of the net with the following pins:

U15.3 U14.3 U16.3 U5.3 U11.2

SIGNAL END1

where END1 is the name of the net with the following pins:

U13.3 U12.3 U18.3 U19.3 U20.3 U21.3

The SPLITNET statement is used in "To Schematic" only. This statement may not be used in an ECO file which will be ported to a PCB design.

5. Rename net name in the design.

The header line is:

RENNET RENAME NET

The format for the rename net information is:

FOO BERF

where foo is the old net name and berf is the new net name.

CLK0 CLKA
PWR VCCD

6. Add part to the design.

The header line is:

PART ADDPART

The format for the part information is:

U1 74LS00

where U1 is the part reference name and 74LS00 is the part type name.

NOTE: When parts are added, they will be placed at system origin of 0,0 and therefore will need to be placed.

7. Delete part from the design.

The header line is:

DELPART DELETE PART

The format for the part information is:

U4 74LS02

where U4 is the part reference name to delete and 74LS02 is the part type name. (part type name is not required).

R1 RES1/4W

All pins of the part to be deleted must already have been disconnected from the connection nets in the design or an error will be reported.

8. Rename part in the design.

The header line is:

RENPART RENAME PART

The format for the rename part information is:

U7 U1

where U7 is the old name and U1 is the new name.

U8 U2
U2 U4

In order to facilitate the renaming operation duplicate name checking is not run until after all rename information has been operated on. This allows the above rename list to run without a conflict over U2 in the example above. If ANY error is encountered NO parts in the list are renamed.

9. Change part's part type.

The header line is:

CHGPART CHANGE PART TYPE OF PART

The format for the change part information is:

U2 7400 7404

where U2 is the reference name of the part to change. 7400 is the old part type and 7404 is the new part type, otherwise an error is reported.

10. Gate Swapping in the design.

The header line is:

SWPGAT

The format for gate swapping information is:

U1.A U4.B

where U1 and U4 are the part reference names and A and B are the gates to swap.

U3.C U6.C
RN10.E RN10.A

Care must be taken to be sure that the part types of the parts whose gates are being swapped are the same and that the gates are swappable between each other.

11. Pin swapping in the design.

The header line is:

SWPPIN

The format for pin swapping information is:

```
U1  1.5  8.9
```

where U1 is the part's reference designation 1.5 8.9 are pin pairs to swap.

Pins can only be swapped between pins within a gate of a part and not between pins of different gates or parts.

Care must be taken to be sure that the pin being swapped are described in the part type description as swappable. If they are not so described, they will still be swapped but a warning message will be reported.

SAMPLE BACK ANNOTATION FILE
\DEV\APPLIC\ECO>

```
*PADS-ECO* ENGINEERING CHANGE ORDER REPORT
*REMARK* -- MYJOB001 -- Tue Aug 25  14:35:50  1987

*NET*
*SIGNAL* I 12
U17.16  U12.16

*JOINNETS*
DATA0 DATA5
*RENNET*
SIGH  SIGHB

*RENNET*
STROBEL CLK0
DATAA DATA0

*DELPIN*
P1.6        ENABLOW
U1.3        ENABLOW
P1.19       FOOBAR
U11.A1      FOOBAR

*PART*   ITEMS
C30 DCAP1
C31 DCAP1
```

```
*DELPIN*
U2.1      ADD0
U2.2      ADD1
U4.2      ADD1
U2.3      DATA0
U2.4      STROBB
U4.4      STROBB
U2.5      GND
U2.6      SIGH
P1.4      SIGH
U2.7      GND
U2.8      CLONE
U4.12     CLONE
U2.9      ZED011
P1.5      ZED011
U2.10     INT3
U2.13     ENABL0
U2.11     VCO
U2.12     GND
U2.14     VCC   *DELPART*
US 7404
*RENPART*
U7 U4
U4 U7
*CHGPART*
U11 7400 7410
*END*           \PADS\FILES>
```

APPENDIX A4

A SUMMARY OF THE PADS LIBRARY CONCEPT

A SUMMARY OF THE PADS-LOGIC LIBRARY ORGANIZATION

NOTE: The following discussion applies to PADS-2000 and PADS-LOGIC when used together.

The creation of Part Types, Part Decals, their storage in the Library and use in a Schematic or Board database is a highly sophisticated sequence involving a number of different routines in the PADS software. A description of each of these routines is provided in complete detail in their dedicated chapters in this Reference Manual. However, their integration into an integrated system is not covered in the respective chapters. This section (Appendix A4) carries the discussion one step further by providing a road map of the interaction and integration of the various routines that comprise the total PADS Library system.

Section I - A SUMMARY - The PADS LIBRARY

Refer to the figure below for a discussion of the PADS Library:

PLIB	
USER Area	STANDARD Area
2-D Lines	2-D Lines
Part Types (Electrical)	Part Types (Electrical)
CAE Decals	CAE Decals
PCB Decals	PCB Decals

1. The PADS Library for both Logic Capture and PCB Design is controlled by the Library Management program called PLIB. To access the library, be in the root directory and in response to C: > respond PLIB <CR>. The response is the display at the CRT of the PLIB Editor, an alphanumeric text editor.

2. The PADS PLIB Library contains two areas, the Users area and the Standard area.

3. The Users area contains those items that have been created by YOU, the user. The Standard area contains those items supplied by CAD Software, Inc.

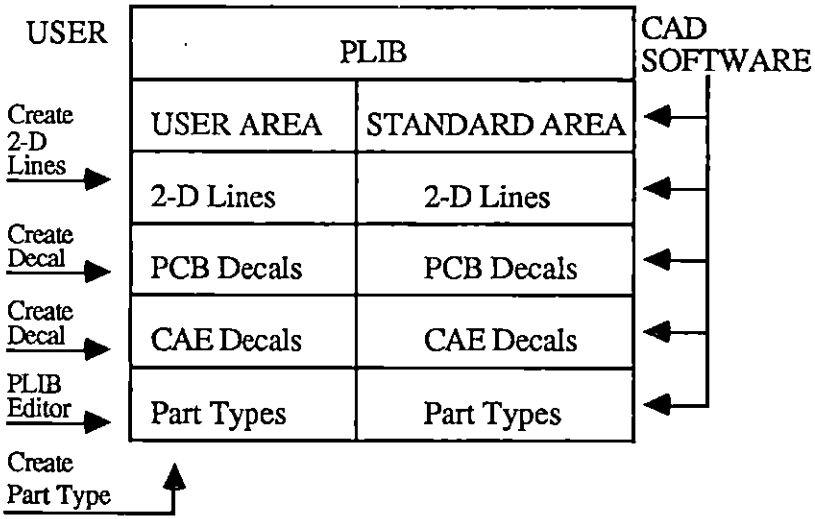
4. The User area contains four categories of item in four sub-libraries: 2-D Lines, CAE Decals, PCB Decals and Part Types (Electrical).

5. The Standard area contains four categories of items in four sub-libraries: 2-D Lines, PCB Decals, CAE Decals and Part Types.

NOTE: The CAE Decals are as used in PADS-LOGIC.

Section II - PLIB CONTENTS GENERATION

Refer to the PLIB Generation figure below during the reading of this section:



1. 2-D Lines, PCB Decals, CAE Decals, and Part Types are stored in the User and Standard areas of PLIB, however, their original generation is from a variety of sources:

2. The 2-D Lines data contained in the Users area is created by the user using the 2-D LINES Routine. The creation is a graphical function using the 2-D Line Graphical Editor, however, the stored item is Text content, and stored in the PLIB 2-D Line Users area.

3. PCB Decals are created as graphical entries in the *PADS- PCB* PART DECAL graphical Editor and normally stored as Text data in the PCB Decal Sub-library of the Users area.

4. CAE Decals and Electrical data are created in the *PADS- LOGIC* Capture system, using the PART LIB Routine.

NOTE: Should you NOT have PADS-LOGIC, Part Electrical data is created using the Text Editor in PLIB, or the CREATE, PART TYPE routine in PADS-2000.

NOTE: While it is possible to use the PLIB Text Editor to modify the Text data associated with 2-D Lines and Decals, this is not recommended. Instead, it is strongly recommended that the 2-D Line and Decal Graphical Editor always be used.

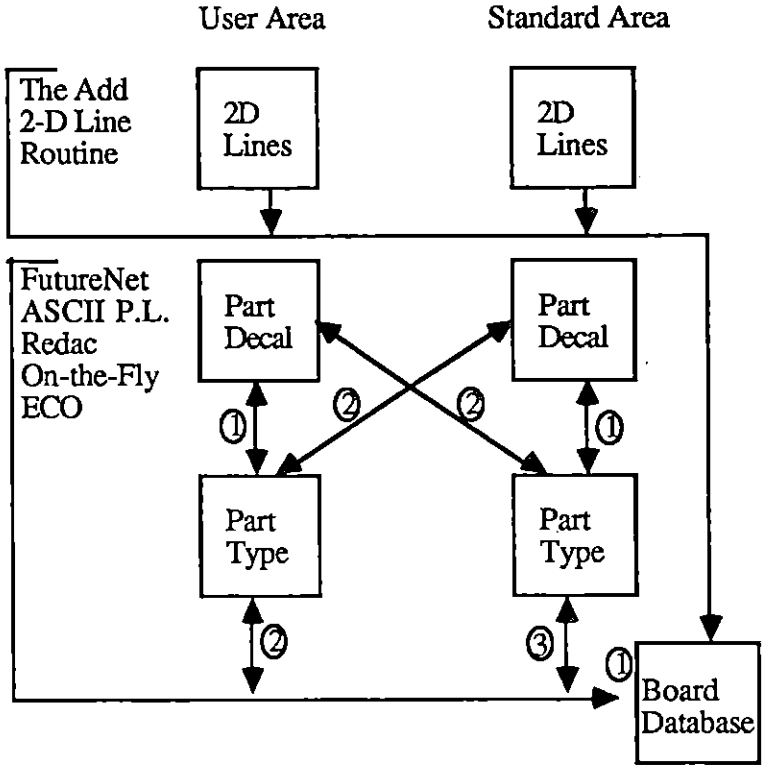
5. The 2-D Lines, Part Decals and Part Types stored in the Standard area of the PLIB Library have been created by CAD Software, Inc.

6. To modify a CAD Software, Inc., supplied 2-D Line or Part Decal stored in the Standard area, the user uses 2-D LINE or PART LIB and modifies these software vendor created items. After modification by the user, they are automatically stored in their original storage area, i.e., the Standard area, unless changed by the user. To store in the User area, SAVE using the name U:XXX where U indicates "to be stored in the User area" and XXX is the name of the item.

7. The user may also modify the Part data supplied by the software vendor and stored in the Standard area. To do so, the user edits the Text data with the PART LIB Routine and saves it in the Standard Part Type area.

SECTION III - USE OF LIBRARY ITEMS IN BOARD DATABASE

1. The retrieval of the three different library items from PLIB and their use in the database of a board is shown in the Figure and described below.



Library Item Retrieval

2. 2-D Lines are retrieved from PLIB and placed in the board database ONLY by the ADD Sub-Routine of CREATE, 2-D LINES Routine. 2-D Line items are NOT added by Parts Lists, Engineering Change Orders (ECO) or by Design-on-the-Fly. When a 2-D item is added to the board by Create 2-D Line - ADD, the User area of the Library is searched first; and if the item is not found, then the Standard area is then searched.

3. Part Types may be added to the board database in a number of ways:

a) ASCII Parts Lists as described in Chapter 13, Volume II.
b) A parts lists contained in the FutureNet In file
c) The parts list contained in the CONV In (Redac) file
d) The parts list contained in the Schema In file
e) The Add Part sub-routine of the ECO Routine
f) The Add Part sub-routine of the Design-on-the-fly Routine

4. When creating a Parts List such as in ASCII, Futurenet, Schema, etc., the Parts List entry MUST have as a minimum, 2 items:

a) the Reference Designation, i.e., U1, R6, C7, etc., followed by:
b) the Part Type name as used in the Part Type Sub-Library of PLIB.

5. When a parts list is read into memory using the Input Routine, the input file is first checked for potential errors including the absence or presence of the part type called for in the Parts List. As the list is read in, the board database is first searched for the presence of each part type. If not present, the search is extended to the User area of the PLIB; if not present, the search is then extended to the Standard area of the PLIB. When the part type is found, the Library Manager then searches for the Part Decal associated with the part type, and when found, the Part Type and its part decal is brought into the board database in memory and displayed at the CRT. If either the part type or the part decal is not found, an error message results.

6. The extraction of a Part Type and its decal using the ECO, ADD PART and the Design-on-the-Fly, ADD PART is the same as for a Parts List; If the part is already on the board, the search process is short in duration, and the part is reproduced instantly, otherwise, a longer search is required of PLIB on the hard disk. This is why during the ECO or Design-on-the-Fly Routine, if the part is not present on the board, the disk library search will take a few seconds longer than if the part were already on the board.

7. Part Decals are extracted from the library and placed in the database of the board after being "pointed" by its associated Part Type. That is, a Part Type must be requested in either a parts list or the ECO/Design-On-The-Fly before its Part Decal is brought into the board database. The relationship between a Part Type and its associated Part Decal is defined by the Part Decal "Pointer" contained in the first line of the Part Type

description. It will be remembered that the first line of the part type file reads:

REF DES PART DECAL TECHNOLOGY, ETC.

When a Part Type is searched for and found, the pointer (the Part Decal name) in the first line of the Part Type file is then used to search the Part Decal sub-library of PLIB for the appropriate Part Decal. This Part Decal is then married with the Part Type and brought into the board database. When the Part Type is in the User area, the Users Part Decal area is first searched for the Part Decal, and if not found, the Standard Part Decal area is then searched. If the Part Type is in the Standard area Part Type area, the Standard area Part Decal is first searched, and if not found, the Users area Part Decal is then searched.

8. It should be seen from the above that a Part Decal can not be ordered directly from the library by the Part Decal name. Instead, its Part Type must always be retrieved first. Attempting to retrieve the Part Decal is the single most common error that users commit in using the PADS Part Library. Remember to always retrieve a part by its Part Type name, NOT its Part Decal name. If in reading in a Part Lists or adding a part with ECO or Design-on-the-Fly, the message "PART NOT FOUND" is present; the most likely reason is that the part was asked for by its Part Decal Name, not its Part Type name.

9. Having stated the above, it is recognized that certain users may want to call-up a Part Decal directly for a special reason. For example, you may want to construct a part list as follows:

U1	Dip14		
U2	Dip14		
U3	Dip20		
U4	Dip28		
Etc	Etc	Etc	Etc

because you have no interest in the Part Type; you merely wish the correct Dip socket. A Design Service Bureau is a good example of such a user situation. The following is a "work around" that you may find useful for such a situation:

1. Create the Part Decal and give it a name like DIP14 for a 14 Pin Dip, etc.

2. Create a Part Type with the Part Type name DIP14. Also, give its Part Decal name the name DIP14 the first line of the part type would read:

```
DIP14  DIP14  TTL  0000.
```

3. Create the Parts List, thus:

```
U1     DIP14
U2     DIP14
Etc    Etc
```

The result of the above workaround is that the Parts List will cause the system to search for the Part Type name DIP14 for U1. When found, the Part Type pointer will then search for a Part Decal called DIP14. With this workaround procedure, you can use Decal names in the Parts List. Be aware, however, that your board database contains no intelligence as to Part Type!

ALTERNATE METHOD OF EXTRACTING A PART DECAL

The Parts List may be used as an alternate method of overwriting the PADS Library Manager and extracting a Part Decal as follows:

Instead of the format:

```
REF DES    PART TYPE
REF DES    PART TYPE
etc.
```

construct the Parts List as follows:

```
REF DES    PART TYPE @DECAL NAME
REF DES    PART TYPE @DECAL NAME
```

The using of @ DECAL NAME overwrites the Part Type and the pointer that searches for the Decal associated with the Part Type and, instead, brings up the Decal whose Name follows @, and assigns this Decal to the REF DES. No checks are made as to whether the Decal is correct for the Part Type, therefore, use this Alternate with caution!

APPENDIX A5

NETWORKING PADS

Users of Multi-station of PADS-2000 may want to network all stations together into a PADS Network. For example, it is easier to maintain control over the Library used by all users, if the Parts Library is under the direct control of the Systems Manager. This control is possible when PADS-2000 is networked, as described in this Appendix.

Should the User wish to network LOGIC, we suggest the use of the NOVELL Network, as produced by Novell, Inc., of 122 East, 1700 South, Provo, Utah 84601.

For illustration purposes, assume a five (5) Terminals, PADS Network. Terminals A, B, C & D are Design Stations. Terminal E is both a Design Station and the Systems Manager Terminal where the System Library is maintained. All parts used in Design on any of the Five Design Terminals are to be obtained from the Systems Manager Library on Terminal E. All parts created or modified on any of the Five Design Terminals are to be stored in the Systems Manager Library. To implement such an arrangement, proceed as follows:

1) Implement the network, using NOVELL. Once the network has been implemented, verify files can be transferred between stations. Transfer ASCII and Job files to and from each station. When satisfied these files are transferrable, proceed with Step 2.

2) Set all library files as shareable READ/WRITE. Use the NETWORK utility "flag" to examine and set the files. In the example shown below, "X:>" equals the network drive.

To set up the library files, proceed as follows:

```
X:> CD \PADS\LIB
X:\PADS\LIB>FLAG *.IDX SRW
```

```
X:\PADS\LIB>FLAG  *.DAT SRW
```

To examine the library files, type the command: FLAG

```
X:\PADS\LIB>FLAG  <CR>

SPART.DAT     SHAREABLE READ WRITE
SPART.IDX     SHAREABLE READ WRITE
ETC.,.
(where FILENAME  NETWORK_ATTRIBUTE)
```

All systems on the network must run "SHARE" (share.exe), this is part of MS-DOS.

```
c:>SHARE
```

3) On terminals A, B, C, and D, (But not E, the Systems Manager Terminal), edit the library manager files PLIB.STD and PLIB.USR to reflect the System Manager terminal. These edits are changes to the Terminal directory path. By modifying these paths, terminals A, B, C, and D will obtain library parts from the System Managers' library on Terminal E. To edit these files, proceed as follows:

Be in the root Directory with C: > displayed at the CRT, using the command:

```
PEDIT \PADS\LIB\PLIB.USR <cr>
```

The PLIB.USR file should now be displayed on your CRT:

```
4444
0  \PADS\LIB\ULINE.DAT          XXX etc.
 4 \PADS\LIB\ULINE.1DX          XXX etc.
  0 20 0

1  \PADS\LIB\UPCBDEC.DAT        XXX etc.
 5 \PADS\LIB\UPCBDEC.1DX        XXX etc.
  0 17 0

2  \PADS\LIB\UCAEDEC.DAT        XXX etc.
 6 \PADS\LIB\UCAEDEC.IDX        XXX etc.
  0 17 0
```

```
3 \PADS\LIB\UPART.DAT          XXX etc.
 7\PADS\LIB\UPART.IDX          XXX etc.
  0 17 0
```

Note that there are eight lines of data in the file, two lines of data for 2-D Lines, two lines of data for Decals, two Lines of data for CAE Decals, and two lines of data for Part Type data.

Using the PEDIT Editor, place an E: at the beginning of EACH of EIGHT lines of data, such that they appear as follows:

```
0  E:\PADS\LIB\ULINE.DAT       XXX etc.
 4 E:\PADS\LIB\ULINE.IDX       XXX etc.
   0 20 0

1  E:\PADS\LIB\UPCBDEC.DAT     XXX etc.
 S E:\PADS\LIB\UPCBDEC.IDX

   etc.    etc.
   etc.    etc.
   etc.
```

After this is done, save the file by overwriting the existing file.

Next, change the standard PLIB.STD file, by using the PEDIT Editor and the command:

PEDIT \ PADS\LIB\PLIB.STD <cr>

Call up the PLIB.STD file. It will appear, as follows:

```
0 \PADS\LIB\SLINE.DAT      etc. etc.
 4\PADS\LIB\SLINE.IDX      etc. etc.
   0 20 0

1  \PADS\LIB\SPCBDEC.DAT        etc. etc.
 5\PADS\LIB\SPCBDEC.IDX         etc.
   0 17 0
     etc.
     etc.
     etc.
```

for eight data lines.

Edit the file by placing an E: at the beginning of each of the eight lines of data, as in the User File above, and save the file by overwriting the Existing file.

3) Repeat the above Editing on terminal A, B, C & D. Test each Terminal Station to insure PADS created parts at the station are stored on Terminal E, the System Manager's Library, and can be retrieved from that Terminal.

NOTES:

(1) While the NOVELL Network is recommended, do not use the NOVELL ARCNET card if planning to use PADS-SUPER ROUTER.

(2) Users should avoid performing simultaneous writes or updates to the library.

(3) Library initialization should not be attempted across the network, and should only be attempted on the local system.

APPENDIX A6

SYSTEM FILES

This appendix provides background information on various files that the user may have occasion to use.

STARTUPL.ASC

This file is used to set the default level of general board parameters such as grid, colors, level, etc.,.) When updating your system to a major software release, it is recommended this file be transferred in a binary format (job file). This is because most PADS ASCII files are nontransferable between one major release to another major release; however the binary files can be transferred. When they are transferred, a transparent conversion program is run, converting the job file to the new releases' format. To accomplish this, simply save a blank screen (that's right, no job active), as startup.job. Once the new software has been installed, read the "startup.job" into the new version. Upon completion of job in, ASCII out the job file using the following steps:

ASCII OUT; SELECT ALL; SELECT GO; OUTPUT
FILENAME: \PADS\ STARTUPL.ASC ; C/R

This now replaces the default startupl.asc with your startupl.asc.

The STARTUPL.ASC file resides in the \PADS Directory.

DRILL.DAT:

This file contains all of the formatting options required for the NC drill output. This file is used only by those users who have the PADS-DRILL option. It will only require archiving if the user has the PADS-DRILL option, and it has been modified. For additional information regarding the DRILL.DAT, refer to Volume II, PADS-2000 Reference Manual, Chapter 11.

The DRILL.DAT file resides in the \PADS Directory.

2000.BAT:

This file executes the PADS-SHELL. It contains path information and can control a 386 memory manager. (refer to Pinstall, the pads installation procedure). If this file has been edited by the user, it is recommended that it be archived on a floppy diskette. If it is not archived, when a new revision of software is installed, the existing 2000.BAT is overwritten.

The 2000.BAT resides in the \PADS Directory.

S*.DAT and S*.IDX LIBRARY FILES:

These binary files contain all of the PADS standard library information. There are a total of eight (8) standard library files. These need only be archived when a user has modified the standard library. The standard library will only be overwritten if the user elects to install the library diskette supplied with new software. (The user library will not be overwritten).

Transfer of individual library items (i.e. DIP14) may be achieved by utilizing the WRITE and READ feature within PLIB. This procedure is outlined in Chapter 12; System Parts Library.

All library files reside in the \PADS\LIB Directory.

PLIB.STD, PLIB.USR: (For more information refer to Appendix A5 - Networking PADS)

The purpose of these two files is for users with networking who wish to have their library in one location, such as a file server. It is through these two files the direction of the libraries can be controlled. They need only be archived by users who take advantage of networking. These files reside in the \PADS\ LIB Directory

BATCH.DAT and DEFAULTL.PAD:

The BATCH.DAT file contains general CAM information such as the drill legend and aperture table. This file is created the first time the user specifies a CAM subdirectory. It retrieves it's initial information from the DEFAULTL.PAD. The DEFAULTL.PAD is non-transferable between revisions of software and will be overwritten when an upgrade is installed.

The BATCH.DAT is transferable between revisions of software and will not be overwritten when a software upgrade is installed. This is because the BATCH.DAT is a file the user (indirectly) creates.

An example for archiving the BATCH.DAT would be the situation of multi-stations or users. The aperture table which was used on job X is needed on Job Y. By copying the BATCH.DAT file into the Job Y CAM subdirectory, the user has avoided the time consuming process of creating the aperture table.

The DEFAULTL.PAD resides in the \PADS Directory.
The BATCH.DAT resides in \PADS\CAM\USERDIR Directory.

APPENDIX A7

ERROR MESSAGES

1. INTRODUCTION:

This Appendix lists and describes the Error Messages that may be generated by the system, at various stages of the design.

In instances where the error message indicates a limit has been exceeded within the job database, it is due to insufficient memory. If the user adds more RAM, the limits will be increased. However, limits within Library parts/decals are hard coded.

386 IS CURRENTLY IN VIRTUAL MODE UNDER ANOTHER PROGRAM

This error message appears when selecting "SROUTE" on a 386 machine. The user has a memory manager active (i.e., 386MAX) which is stuck in expanded mode. The Superrouter looks for extended memory on a 386. Therefore, they must deactivate the manager. To accomplish this:

(a) Some managers have a ".com" file that allows the user to toggle between expanded and extended memory. For instance, 386max will except the statement "386max on" or "386max off" "on" activates the expanded memory, off deactivates the expanded memory, returning it to extended. It is possible to place these statements in the PADS.BAT file. If the user utilizes the shell, the memory will toggle automatically.

(b) Another (but not good) solution: Remove the memory manager from the config.sys and reboot the system. Both the PS2 and Tandy systems at the date of this document, required this procedure, a toggle function was unavailable.

BAD ATTEMPT TO CLOSE DATABASE

User did not initialize the USER library. This is on new installs only. Action: Initialize the library.

BAD PART TYPE PIN NUMBER 74ALS29818\DW 28

The part type file defines a 28 pin device, yet the decal defines a 24 pin device. Action: In this case the decal is incorrect--change the decal.

CAMDVR: FATAL ERROR - CAN'T ACCESS FONT FILE

Somehow the CAMFONT file became corrupted in the PADS directory. Action: Reload software.

CAN'T COMBINE ANY MORE DRAWING ITEMS INTO THIS ONE (arcs limit exceeded)

Appears in 2D line, or decal creation.

CAN'T COMBINE ANY MORE DRAWING ITEMS INTO THIS ONE (corner limit exceeded)

Appears in 2D line, or decal creation. (Please note a line segments' end points equal two corners).

CAN'T COMBINE ANY MORE DRAWING ITEMS INTO THIS ONE (strings limit exceeded)

Appears in 2D line, or decal creation. The maximum text allowed per string is 80.

CAN'T COMBINE ANY MORE DRAWING ITEMS INTO THIS ONE (pieces limit exceeded)

Appears in 2D line or decal creation. The definition of a piece is a line element created using new poly/end, or new poly/complete. Each whole element constitutes a piece; a circle is one piece, an arc is one piece.

CAN'T DECREASE MAXIMUM NUMBER OF LEVELS

Layers are easily increased, however, not decreased. To decrease: have job active; ascii out all; edit ascii file, max layer statement, changing to new number. Ascii file in to a blank screen, save as a new job.

CAN'T FIND DECAL ITEM

Loading a part type where the decal does not exist. Action: Create the decal using CREATE/PART DECAL.

CAN'T FIND PART TYPE ITEM

Loading a part type where the part type does not exist. Action: create the part type using PLIB/USER / PARTTYPE.

CAN'T FIND PLIB.USR

Check to see if the Plib.usr exists in the \LIB directory. If the software was recently loaded, it is possible the LIB.ARC did not unpack (not enough disk space). Action: Reload library., insure there is enough disk space.

CAN'T FIND ULINE.DAT

(1) The User library was never initialized. Action: enter PLIB, initialize the user library.

(2) DOS is installed more than once on the system. Action: Remove one DOS, or remove path to one DOS.

CAN'T OPEN TEMPORARY CHECKING FILE \PADS CHK 12.TMP 23

(1) Too many layers to check at a time. Action: Check one layer at a time. To achieve this, display pads / track / copper, lines etc., one level at a time.

(2) Not enough disk space. Action: clean up hard disk deleting any unnecessary files.

(3) FILES=20 is missing from CONFIG.SYS. Action: Add the statement files=20 to config.sys (reboot).

CAN'T ROUTE FROM INVISIBLE NET XXX

Occurs when user has Net Attribute Display turned OFF. Turn Net ON in NET ATTR Display.

CAN'T OPEN TEMPORARY CHECKING FILE 28

(1) Too many layers to check at a time. Action: Check one layer at a time. To achieve this, display pads, track / copper, lines etc., one level at a time.

(2) Not enough disk space. Action: Clean up hard disk, deleting any unnecessary files.

(3) FILES=20 is missing from CONFIG.SYS. Action: Add the statement: files=20 to config.sys (reboot).

CONNECTION EXCEEDS WRONG WAY LIMIT

The definition of a wrong way is: The non-preferred routing direction (i.e., on a horizontal layer the wrong way is vertical). This message indicates the wrong way is too large. All of the non-via passes will support a wrong way limit of 100 mils, and a maximum track width of 15 mils. Different passes of the autorouter have specific parameters, these are:

Short memory: 700 mils between pin pair max; connection must be a straight horizontal or vertical, no diagonals.

Long memory: 1500 mils between pin pair max; connection must be a straight horizontal or vertical, no diagonals.

Horiz/Vert: 800 mils between pin pair max; connection may be diagonal of a max of 100 mils.

Short only: 1000 mils between pin pair max; connection may be diagonal of a max of 100 mils.

Long Only: 31000 mils between pin pair max; connection may be diagonal of a max of 100 mils.

There is a file in the PADS directory "RouteXX.dat" (where xx is routing grid). This is an ascii file which may be edited by the user, however, CAD Software does not recommend editing. If there are any questions regarding this file, please contact Cad Software Technical staff.

ERROR CODE 1690

No board outline. Action: Using the CREATE; BOARD; NEW POLY; Create a board outline.

ERROR CODE 3659

Insufficient disk space. Action: Clean up the hard disk.

ERROR CODE 3950

Insufficient disk space. Action: Clean up the hard disk.

ERROR CODE 3994 - - Error accessing internal database

A component pad is on top of another component pad of the same net. Action: Scan the board for two component pads on top of one another, or run CHECK; move one of the components.

ERROR CODE 1137

Via on the edge of the board. Action: Scan the board, move the via.

ERROR CODE 21831

Not a valid routing grid. action: Change the routing grid. Allowable PADS-ROUTE grids are: 10, 20, 25, 50.

ERROR CODE 22670

Not a valid routing grid. Action: Change the routing grid. Allowable PADS-ROUTE grids are: 10, 20, 25, 50.

ERROR CODE 24138

Not a valid routing grid. Action: Change the routing grid. Allowable PADS-ROUTE grids are: 10, 20, 25, 50.

ERROR LOADING DATABASE, FUNCTION R_BORDER, ILLEGAL CALL SEQUENCE PROGRAM STOPPED

User is not utilizing the PADS shell and is accessing Superrouter thru a sub-directory. Action: Use shell, do not use subdirectory other than those created by PADS.

FILE ACCESS DENIED (may be write protected)

Occurs during backup of a file to floppy or hard disk. Make sure the media the file is being sent to is not write protected. Try another disk.

FILE I/O ERROR READING ROUTER.DAT ERROR CODE - 12

Not a valid routing grid. Action: Change the routing grid. Allowable PADS-ROUTE grids are: 10, 20, 25, 50.

INSUFFICIENT MEMORY AVAILABLE, USE WINDOW MODE TO REDUCE ROUTING AREA

Not enough memory to route entire board, action:

(1) Insure there is a board outline. If there is none, create one.

(2) Check for resident memory programs, remove, and reboot.

(3) Board area is too large; use box window (keypad del), window around area to be routed. Select WINDOW in the autorouter menu.

(4) If there is a board outline, it could be too large for the amount of memory on system.

INTERNAL PROGRAM ERROR, R_ROUTE . . .

Not enough memory available. Action: Insure the board is set for the correct "maximum routing levels;" memory is required for each routing

level. This max level should not include embedded planes. Determine how much memory is required for the design (refer to Superrouter appendix in PADS reference manual). Insure no other programs, including networking, are not resident in memory.

I/O ERROR WRITING FILE, FILENAME, ERROR XXX

Occurs during backup of job to hard disk or floppy, or not enough space on hard disk. Check appropriate drive to ensure it is ready.

LAYER IS NOT A VALID ROUTING LAYER

Trying to route on a layer which is not defined as a MAXLAYER. Action: change level for routing; or increase max layer.

LINES ITEM NAME TOO LONG

Maximum character length for lines = 16

LINES WIDTH EXCEEDS MEMORY LIMITS

Trying to route a track greater than what is supported in the autorouter. All non via passes will route a connection no greater than 15 mils. Anything larger is considered a PWR/GND route.

LOADING DATABASE FUNCTION R_PLCFAREA, . . .

Occurs at start of route within Superrouter. Action: There is copper placed on layer 0. Remove from layer 0 and place on appropriate routing levels.

NEED MORE MEMORY SPACE - - - JOB FILE WAS CREATED ON A SYSTEM WITH MORE MEMORY

Action:

(1) Check system job was originally created on, run job limits; compare memory. Add more RAM.

(2) User could have <CNTRL C> out of a program, thus leaving program resident in memory. Reboot system.

NO MORE ROOM IN DATABASE FOR THIS KIND OF ITEM (This message varies depending on item)

Reached system limits. Action: Run job limits report to determine which item. Remove any resident memory programs, reboot. Add more memory

NOT ENOUGH DISK SPACE FOR FILE, FILENAME.EXT (TRY ANOTHER DRIVE)

Insufficient disk space on floppy or hard drive. Try another floppy, or a separate drive on hard disk.

NO SUCH DIRECTORY OR CAN'T CREATE DIRECTORY

Occurs when a user is archiving to another directory, specifying the incorrect path. An example is as follows:

A:USERDIR\MYJOB.JOB

should be:

A:\USERDIR\MYJOB.JOB

OUT OF ENVIRONMENT SPACE

Path too long. Too many set commands in AUTOEXEC.BAT. Action: Remove set commands, shorten path, reboot.

PAD DOES NOT EXIST ON SELECTED LAYER

User is routing to a pad which is not defined for the active layer. (i.e., four layer board, routing on layer two, pads are only defined on layer one and four.) Action: Redefine pad stacks, specify inner layers.

PAD STACK PIN NUMBER 60 NOT IN SYMBOL XXX

Defining pad stack information BEFORE terminals are placed on screen. Action: Start over, placing terminals first, then define pad stacks.

PART TYPE NAME TOO LONG

Maximum character length of a parttype = 16.

PLOT IS TOO LARGE

The area to be plotted is too large for the photoplotter. This dimension is determined by the outmost graphics, i.e., moires, dimensions, notes --- not just the board outline. Whether the user has selected margins, centering, or origin offsets, if any graphics is outside the plotting area, this message will appear. Action: Determine actual plotting area of job. The plotting area is determined in PINSTALL, the defaults are:

Default for Gerber = 16000 X 20000

Default for Laser = 42000 X 32000

To change the defaults, execute pinstall, skipping software installation (answer no to software install), proceed with hardware installation. Change X and Y dimension to appropriate plotting bed size.

PPBATCH: FATAL ERROR WHILE ACCESSING BATCH FILE

The Defaults.pad is missing from the PADS directory. Action: Copy DEFAULTL.PAD from original software diskette into PADS directory, reinstall hardware using pinstall procedure, hardware setup only.

PPBATCH: UNABLE TO OPEN DEFAULTS FILE

(1) There are two versions of pads installed on system. Delete one, or rename the directory.

(2) The wrong revision of PINSTALL was used to install the software. Action: Reinstall the software using the appropriate revision of PINSTALL.

NOTE: It is common for a user to copy PINSTALL onto the hard disk. Insure when a new revision of software is installed, that the new pinstall is installed at the same time.

PPUSER: FATAL ERROR - UNABLE TO CREATE SUB-DIRECTORY

The directory \PADS\CAM\ does not exist. Action: Exit software, create a CAM directory. (Cause: User "copied" software instead of installing).

PP?????: FATAL ERROR, UNABLE TO ACCESS IF?

This first statement varies depending upon what command is being executed. Generally speaking, any time the above message appears, check the following:

(1) Insufficient disk space. Action: Clean up the hard disk.

(2) Insure FILES=20 appears in the config.sys.

(3) No other programs are resident in memory (check config.sys and autoexec.bat); reboot.

(4) Check the stability of the system.

(5) Reload software.

PPXTRCT: FATAL ERROR, UNABLE TO ACCESS IF1

Insufficient disk space. Action clean up the hard disk.

SECTOR NOT FOUND WRITING DRIVE C: ABORT, RETRY, FAIL?

The DOS statement "FASTOPEN" is resident in memory. Action: Remove from config.sys.

SECURITY KEY NOT FOUND . . . EXITING . . . . .

(1) CAD Software security key is missing from the parallel port.

(2) Security key has a printer cable which is not fully wired, plugged into it, or a bad printer cable. Test by removing printer cable from key.

SHORT TO PIN

Occurs during route within Superrouter. Action: Insure there are no violations. Check component pad size, spacing, track width. (1/2 track + 1/2 space + 1/2 routing grid = or less than multiple of routing grid. A situation in which this message occurs is when the pitch on a component is 25 mils, routing grid = 10 mils, tracks and space are 5 mils each. The router will round to the next available ROUTING GRID. The first grid is taken 1/2 the component pad + space. The next grid is taken by the abutting component pad, therefore, the "short to pin." (Using the formula above 1/2T+1/2S+1/2G=11; half of 5 is not easily divided by the router, so it rounds up 3+3+5). In order to route on the Superrouter, a 16 mil grid must be used.

SIGNAL NAME TOO LONG

Maximum character length = 12.

STANDARD VIA SHAPE NOT ROUND OR NOT SAME SIZE ON BOTH LAYER OF LAYER PAIR

Via is not defined on all routing layers. Via is not the same size or shape on all layers. Action: Change pad stack data for via; Go to setup; pads; input: standardvia.

UNABLE TO ACCESS IF2

Not enough disk space. Files statement missing from config.sys (FILES=20).

UNABLE TO FIND OPEN CHANNEL FOR ROUTE

The Router parameters have been exceeded. Something is blocking the router. Action: Check pad size, spacing. General rule: 1/2 track + 1/2 space + 1/2 routing grid = routing grid or multiple thereof.

UNABLE TO OPEN GERBER FILE

Not enough disk space. Files statement missing from config.sys (FILES=20). Path is incorrect.

WRITE FAULT ERROR WRITING DEVICE. . .

This is a DOS error message. It can appear upon output to the penplotter, or any other output device. This error message results upon output to the penplotter when there is a poor connection in either the cable or port. It can also occur when the mode command is not being implemented correctly. (MODE COM?:9600,N,8,1,P).

CRASH

Look for resident memory programs:

"LIGHTENING" - is a program to accelerate system; appears in config or autoexec.

"FASTOPEN" - A DOS statement which is automatically installed in Version 4.0, available in previous revisions of DOS.

EXITS PROGRAM

PADS-2000:

- Security key has been removed from parallel port.

PADS-2000 SUPERROUTER:

- Security key has been removed from parallel port.
- Security key has not been programmed for Superrouter.

GENERAL CHECKLIST WHEN THE PENPLOTTER IS NOT FUNCTIONING:

Insure DOS is loaded and the PATH set to DOS in the Autoexec.bat. Insure the PATH statement appears prior to the MODE statement in the Autoexec.bat.

Insure the statement "MODE COM2:9600,N,8,1,P" has the "P" for pause in the Autoexec.bat.

Insure there isn't a batch file prior to the MODE command in the Autoexec.bat. If there is, it could be preventing the MODE command from being implemented.

Check PINSTALL; Insure the correct plotter is selected, the correct paper size is selected, (mils not inches), the plotter is set for the correct port and is on line.

It is not possible to share the com port with another device - - such as the mouse. If only one port is available, the plot file must be created off-line, then copied to the com port. Prior to copying to the com port, the MODE command must be implemented. To accomplish this, at the following prompt:

C:\PADS\CAM\USERDIR

TYPE:

MODE COM1:9600,N,8,1,P <CR>

The system should echo back "Resident portion of MODE loaded COM1:9600,N,8,1,P"

To output the file to the com port, type the following:

COPY FILENAME.EXT COM1: <CR>

APPENDIX A8

(PADS-PCIF)

P-CAD™ INTERFACE

1.0 GENERAL DESCRIPTION

PADS-PCIF is a bidirectional translation program that can be used to convert a P-CAD PDIF OUT file to a PADS-2000 (Version 3.0 or greater) or PADS-PCB ASCII IN file.

PADS-PCIF can also convert a PADS-2000 or PADS-PCB ASCII OUT file to a P-CAD PDIF IN file.

2.0 PADS-PCIF SETUP

A Setup file is required to define the conversion conventions required to translate P-CAD PDIF OUT data to PADS-2000 or PADS-PCB ASCII IN data.

2.1 PREPARING THE SETUP FILE

The PADS-PCIF program requires that the user provide a guide for the assignment of P-CAD named layers to the corresponding PADS-2000 or PADS-PCB layer numbers. This is accomplished through the use of a set-up (.STP) file. This file can be produced and edited with most text editors, or it may be semi automatically generated through the use of /PPADS.EXE. (Optional see Section 6.0).

*NOTE: This file is only needed when converting from P-CAD to PADS-2000 or PADS-PCB.

The following is an "actual" setup file. It is used to explain the setup file format.

FORMAT EXAMPLE

1. ; setup file for PADS-PCIF
2. X indicates a flag for the PADS-PCIF file
3. ***** L "layer name in PADS-PCIF"
4. L "COMP" 1

5. ***** PAD 0 is used for vias and MUST be in the setup file
6. P 0 -2 50 R 35
7. P 0 -1 50 R
8. P 0 -0 50 R
9. ; END

NOTES:

A. Statements preceded by a semicolon (;) are treated as comments, and are not seen by PADS-PCIF.

B. Statements containing five asterisks (*****) indicate data instructions.

C. (X) indicates a flag which will appear when executing PADS-PCIF.

D. Statements preceded by an (L) provide the P-CAD to PADS layer assignment data. The P-CAD layer name is entered first in quotation marks, then the PADS-2000 or PADS-PCB layer number is entered. Each field is separated by a space.

E. Statements preceded by (P) and a space, followed by (0) zero, indicate the via format.

FORMAT EXAMPLE LINE DESCRIPTION

1. SETUP file comment (Note A)
2. This comment indicates the usage of (X) as a flag (Note C)
3. This statement is instructing PADS-PCIF on the layer names to layer numbers conversion (Note E)
4. This instruction states that "COMP" (Component side) is to be assigned to Layer 1 (Note D)
5. This statement is instructing PADS-PCIF on the via PAD information (Note E)

6-8. P-CAD via pin type to PADS-2000 or PADS-PCB via pad size translation. (Must be in setup file) (Note E)

9. End of file.

3.0 PADS-PCIF PAD FILE

A PAD (.PAD) file is required to define the PAD conversion conventions needed to translate P-CAD PDIF Out data to PADS-2000 or PADS-PCB ASCII IN data.

3.1 PREPARING THE PAD FILE

The PADS-PCIF program requires that the user provide a guide for the assignment of P-CAD pin type data to the corresponding PADS-2000 or PADS-PCB pad stack format. This is accomplished through the use of a PAD (.PAD) file. This file can be produced and edited with most text editors, or it can be semi automatically generated through the use of PPADS.EXE (Optional see Section 6.0).

NOTE: This file is only needed when converting from P-CAD to PADS-PCB.

The following is an "actual" PAD file. It is used to explain the PAD file format.

FORMAT EXAMPLE

```
1. ; PAD file for PADS-PCIF
2. P0 -2 50 R 35
3. P0 -1 50 R
4. P0 -0 50 R
5. ; END
```

NOTES:

See section 2.1 (setup file notes) *Notes A&E apply here.

FORMAT EXAMPLE LINE DESCRIPTION

1. PAD file comment (Note A)

2.-4. P-CAD pin type number to PADS-PCB, Pad Size Translation (Note E).
5. END OF FILE

4.0 .SSS FILE DESCRIPTION

There is no way to directly transfer pad information into P-CAD, since P-CAD keeps all pad information separate from the design file as padstack information. PADS-PCIF permits the user to associate pin type information in P-CAD with the appropriate padstack information within the PADS-2000 or PADS-PCB design. The .SSS file has the same name as the PDIF file with the .SSS extension. This file contains lines of comment information that describe the pad information for each pin type in P-CAD, followed by the padstacks for each PADS-2000 or PADS-PCB pad size. This file is generated automatically by our interface, and is used to manually update the P-CAD GSSF file.

5.0 INITIATING PADS-PCIF

Change to the subdirectory containing the PADS-PCIF program.

Enter the following at the DOS prompt:

PDIFPADS <ENTER>

NOTE: Typing "PDIF PADS" allows a conversion from P-CAD to PADS-PCB. Typing "PADS PDIF" allows a conversion from PADS-2000 or PADS-PCB to P-CAD.

The program will display the basic working screen that identifies the program and version number. Pressing the <ENTER> key changes the screen to a line entitled "filename" and the following prompt:

INPUT FILE NAME [.PDF]:
([.ASC])

Enter the file name of the P-CAD PDIF file, including the path, if any, but DO NOT INCLUDE THE .PDF EXTENSION, as this will automatically be added by the PADS-PCIF program.
Pressing the <ENTER> key changes the screen to a line entitled, "Change .STP and .PAD Filename [PDIFPADS], and the following prompt:

INPUT NEW NAME [NO EXTENSION]:

NOTE: This screen will not be present when converting from PADS-2000 or PADS-PCB to P-CAD.

At this point, you can tell the PADS-PCIF program the name of your Setup and Pad file, or you can depress the <ENTER> key to utilize the default file names, (PDIFPADS.STP, PDIFPADS.PAD).

Next, you will be asked if you want the routed tracks to be converted. If you wish all of the routed tracks to be converted, enter:

"Y" <ENTER>

If you do not want the routed tracks to be converted, enter:

"N" <ENTER>

The conversion process begins and various phases will be identified at the CRT screen.

A PADS-2000 or PADS-PCB ASCII IN file will be created as converted output from the P-CAD PDIFOUT file and entered into the PADS-2000 or PADS-PCB program. This .ASC file contains all of thePADS-2000 or PADS-PCB printed circuit board design information. The file is automatically assigned the original .PDF input file name with the exception of the .ASC extension. The ASCII file must be copied to the \PADS\FILES directory and read into PADS via ASCII IN.

NOTE: .PDF files must be read into P-CAD via PDIF IN.

6.0 PPADS

General Description:

PPADS is a semiautomatic setup file and pad file generator that is used when converting P-CAD PDIFOUT files to PADS-2000 or PADS-PCB ASCII IN files.

Key Pad Commands:

The PPADS Editor consists of four keystrokes. The Up and Down arrow keys which are used for moving through the displayed information; Enter is used to invoke the Edit mode on the line that is highlighted; and F1 is used to proceed to the next screen or End.

6.1 Initiating PPADS

From the /PADSPCIF directory, type:

PPADS <ENTER>

Input filename [.PDF]

Enter the filename of the P-CAD PDIF file that you wish to convert, including a path if any.

The screen changes to a list of the P-CAD layer names used in the input .PDF file.

Option 1: Use PPADS internal editor to create your Pad and Setup file.

Option 2: Use the DOS print screen utility to print layer names and pin type information that is produced by PPADS which can be used to manually update your master pad and setup file with the Editor of your choice.

Using the PPADS Editor, assign the PADS-2000 or PADS-PCB Layer Number to its corresponding P-CAD Layer Name. Upon completion, select "F1," the CRT changes to the P-CAD pintype/sizes used in the PDIF file. Selecting <ENTER> will display the PADS-2000 or PADS-PCB padstack data to be updated. Using the PPADS Editor, input the proper PADS-2000 or PADS-PCB padstack data. Once complete, select "F1".

CREATE FILE [FILENAME .STP]

Enter a new filename or select <ENTER> to utilize your input filename with a .STP extension.

CREATE FILE [FILENAME .PAD]

Enter a new pad filename or select <ENTER> to utilize your input filename with a .PAD extension.

7.0 PADS-PCIF ERRORS

1. Lost in Files - It is possible that if the file format is badly corrupted for some unexplained reason, the PADS-PCIF program can lose its orientation. If this happens, you will be given an error message to call CAD Software for assistance. A copy of the file will probably be needed to investigate the nature of the problem thoroughly.

2. Heap Memory - If there are too many memory resident programs present, PADS-PCIF may not have enough heap memory to successfully execute. If this error message occurs, remove some of the memory resident programs from memory and rerun the program.

3. Disk Full - If the disk onto which the converted files are to be written does not have enough room for the file, this message will be displayed. If this occurs, use another floppy, or make room on your hard disk for the files.

4. Fatal Error - If the Setup file is not found, PADS-PCIF will abort and return you to the DOS prompt. If this occurs, make certain that the Setup file for the PDIF file to be converted has been included in the same directory as the PADS-PCIF executable files, and that it has a .STP extension.

8.0 NOTES

1. PADS-PCIF recognizes a six character limit in PADS-PCB for component names. Names that are longer are automatically shortened (last characters are truncated).

2. The PADS-PCIF program is compatible with Version 2.0 and later versions of P-CAD's PDIF programs. However, if you are using named Copper areas, PADS-2000 or PADS-PCB will not recognize them for connectivity purposes.

3. P-CAD provides a "polyvoid" feature for specifying a clearance in copper areas.

(NOTE: PADS-PCB DOES NOT SUPPORT POLYVOIDS.)

4. Padstacks defined in P-CAD for drill holes must be specified with a width. PADS-2000 or PADS-PCB will not accept a 0 width pad. Use a 1 mil pad in the padstack definition for the PAD file for these padstacks.

5. The via pad for PADS-2000 or PADS-PCB must be defined in the Setup file. NOTE: If the via padstack is used for pads other than Vias, it must also be included in the PAD file.

6. When converting from PADS-2000 or PADS-PCB database to P-CAD, PADS-PCIF automatically assigns the following layer information:

PADS-PCB LAYER	P-CAD LAYER
1	LAY 1
2	LAY 2
3	LAY 3
}	}
29	LAY 29
30	LAY 30

The following layers are pre-assigned:

"PIN"	- Thru hole pins
"PINBOT"	- Surface pads, bottom
"PINTOP"	- Surface pads, top
"SLKBOT"	- Silkscreen (component outlines), bottom
"SLKTOP"	- Silkscreen (component outlines), top
"CMPBOT	- Component reference designators, bottom
"CMPTOP"	- Component reference designators, top

APPENDIX A9

PARTS LIST

AND

DECAL PLOTS

PART I

LIBRARY PARTS

This Appendix contains an Alphanumeric listing and drawings of both the Part Types as well as the Decals shipped with PADS-2000 and PADS-LOGIC. Part I contains the alphanumeric listing of the Part Types. Associated with each Part Type is its PCB Decal and Logic Circuit Symbol as used in PADS-LOGIC. Part II contains Drawings of the PCB Decal. Part III contains drawings of the PADS-LOGIC Circuit Symbol.

CAUTION: The User is cautioned that while CAD Software, Inc. has used due diligence to insure the accuracy of the Parts Library, CAD Software, Inc. will not be responsible for any consequential damages resulting from the use of the library. We recommend that you check the Part Type information and Gate information against the Vendor's specification sheet or other data you may have.

The first column in Part I is an alphanumeric listing of the Part Types in the Library. The second column is the associated PADS-2000 PCB Decal, and the third column is the associated PADS-LOGIC symbol.

Part Type	PADS 2000 Decal	PADS Logic Symbol
$GND_SYMS	PCBNULL	GND: AGND: CHGND
$OSR_SYMS	PCBNULL	REF IN: REFOUT
$PWR_SYMS	PCBNULL	+5V: BUBBLE: Y
100415	DIP16	100415
100469	DIP24\400	100469
100470	DIP18	100470
100474	DIP24\400	100474
100480	DIP20	100480
100P14	DIP20	100P14
100P44	DIP20	100P44
100P88	DIP28\600	100P88
10415	DIP16	100415
10469	DIP24\400	100469
10470	DIP18	100470
10474	DIP24\400	100474
10480	DIP20	10480
10KP14	DIP24\600	100P14
10KP14J	DIP24\600	100P14
10P14	DIP20	100P14
10P44	DIP20	100P44
10P88	DIP28\600	100P88
1372	DIP14	1372
146805E2	DIP40\600	DIP38
146805E2Z	CC40\40	DIP38

Part Type	PADS 2000 Decal	PADS Logic Symbol
146805F2	DIP28\600	DIP26
146805F2-PLCC	CC40\40	DIP26
146805F2Z	CC40\40	DIP26
146805G2	DIP40\600	DIP38
146805G2Z	CC40\40	DIP38
146805H2	DIP40\600	DIP38
146805H2Z	CC40\40	DIP38
146818	DIP24\600	DIP22
146818A	DIP24\600	DIP22
146818Z	DIP24\600	DIP20
146823	DIP40\600: CC40\40	DIP38
1468705F2	DIP28\600	DIP26
1468705G2	DIP40\600	DIP38
16164PLCC	CC68	16164
16450	DIP40\600	8250
16550	DIP40\600	16550
2130	DIP48\600	RS1024K8
2130L	CC52: CL52	RS1024K8
2147	DIP18: CLR18	2147
2148	DIP18: CLR18	2148
2149	DIP18: CLR18	2148
2150	DIP24	2150
2167	DIP20: CLR20	6167
2168	DIP20: CLR20	6168

Part Type	PADS 2000 Decal	PADS Logic Symbol
2169	DIP20: CLR20	6168
21L41	DIP18	21L41
21L50	DIP24	2150
24C16	DIP8	24C16
2641	DIP24\400	2641
2641A	CC28	2641
2651	DIP28\600	2651
2652	DIP40\600	2652
2652A	CC44	2652
2653	DIP16: DIP16\SO	2653
26562	DIP48\600	68562
2661	DIP28\600: CC28	2661
2670	DIP28\600	2670
2671	DIP40\600	DIP38
2672	DIP40\600	2672
2672A	CC44	2672
2673	DIP40\600	DIP38
2674	DIP40\600	2672
2674A	CC44	2672
2675	DIP40\600	2675T
2675T	DIP40\600	2675T
2677	DIP40\600	2677
2677A	CC44	2677
2681A	CC44: CC44A	2681N40

Part Type	PADS 2000 Decal	PADS Logic Symbol
2681N24	DIP24\400	2681N24
2681N28	DIP28\600	2681N28
2681N40	DIP40\600	2681N40
2691	DIP24\300	2691
2691A	CC28	2691
2692A	CC44: CC44A	2681N40
2692N24	DIP24\400	2681N24
2692N28	DIP28\600	2681N28
2692N40	DIP40\600	2681N40
2698	DIP64\900	2698SM
2698A	CC84	2698
27128A	DIP28\600	RE16K8: RE16K8-A
27128AL	CLR32	RE16K8. RE16K8-A
2716B	DIP24\600	RE2K8
27256	DIP28\600	RE32K8: RE32K8-A
27256L	CLR32	RE32K8: RE32K8-A
2732B	DIP24\600	RE4K8
27512	DIP28\600	27512
2764A	DIP28\600	2764
2764AL	CLR32	2764
27C010	DIP32\600: CCR32	RE128K8: RE128K8-A
27C020	DIP32\600	RE256K8: RE256K8-A
27C1023	DIP32\600	RE128K8: RE128K8-A
27C1024	DIP40\600	RE64K16

Part Type	PADS 2000 Decal	PADS Logic Symbol
27C1024L	CL44: CC44	RE64K16
27C128	DIP28\600	RE16K8: RE16K8-A
27C256	DIP28\600	RE32K8: RE32K8-A
27C256L	CLR32	RE32K8: RE32K8-A
27C256OTP	DIP28\600	RE32K8: RE32K8-A
27C512	DIP28\600	RE64K8: RE64K8-A
27C512L	CLR32	RE64K8: RE64K8-A
27C512OTP	DIP28\600	RE64K8: RE64K8-A
27C64	DIP28\600	RE8K8: RE8K8-A
27C64L	CLR32	RE8K8: RE8K8-A
27LS00	DIP16	82LS16
27LS00L	CL20	82LS16
27LS01	DIP16	82LS16
27LS01L	CL20	82LS16
27LS02	DIP16	82S25
27LS02L	CL20	82S25
27LS03	DIP16	82S25
27LS03L	CL20	82S25
27LS06	DIP16	82S25
27LS06L	CL20	82S25
27LS07	DIP16	82S25
27LS07L	CL20	82S25
27LS18	DIP16	82S123
27LS185	DIP18\300	82S185

Part Type	PADS 2000 Decal	PADS Logic Symbol
27LS185L	CLR28: CL28	82S185
27LS18L	CL20	82S123
27LS19	DIP16	82S123
27LS191	DIP24\600	27S190
27LS191L	CLR32	27S190
27LS191S	CL28	27S190
27LS19L	CC20: CC20A: CLR20	82S123
27LS291	DIP24\600: DIP24\300	27S190
27LS291L	CLR32	27S190
27LS291S	CL28	27S190
27PS181	DIP24\600	82S181
27PS181L	CLR32	82S181
27PS181S	CL28	82S181
27PS185	DIP18	82S185
27PS185L	CLR28: CL28	RE2048K4
27PS191	DIP24\600	27S190
27PS191L	CLR32	27S190
27PS191S	CL28	27S190
27PS191U	CLR32	82S191
27PS281	DIP24\600: DIP24\300	27S180
27PS281L	CLR32	82S181
27PS281S	CL28: CC28	82S181
27PS291	DIP24\600	27S190
27PS291L	CLR32	27S190

Part Type	PADS 2000 Decal	PADS Logic Symbol
27PS291S	CL28	27S190
27PS291U	CLR32	82S191
27PS41	DIP20	82HS195
27PS41J	CC28	82HS195
27S12	DIP16	82S130
27S12L	CL20	82S130
27S13	DIP16	82S130
27S13L	CL20: CC20: CC20A	82S130
27S15	DIP24\600	82S115
27S180	DIP24\600	27S180
27S180L	CLR32	82S181
27S180S	CL28	82S181
27S181	DIP24\600	27S180
27S181L	CLR32	82S181
27S181S	CL28: CC28	82S181
27S184	DIP18	RE2048K4
27S184L	CLR28	RE2048K4
27S184S	CL28	RE2048K4
27S185	DIP18	RE2048K4
27S185L	CLR28	RE2048K4
27S185S	CL28	RE2048K4
27S190	DIP24\600	27S190
27S190L	CLR32	27S190
27S190S	CL28	27S190

Part Type	PADS 2000 Decal	PADS Logic Symbol
27S191	DIP24\600	27S190
27S191L	CLR32	27S190
27S191S	CL28	27S190
27S20	DIP16	82S126
27S20L	CL20	82S126
27S21	DIP16	82S126
27S21L	CL20	82S126
27S25	DIP24	27S25
27S25L	CLR32	27S25
27S25S	CL28	27S25
27S27	DIP22	27S27
27S28	DIP20: CL20	27S28
27S280	DIP24	27S180
27S280L	CLR32	82S181
27S280S	CL28: CC28	82S181
27S281	DIP24\600: DIP24	27S180
27S281L	CLR32	82S181
27S281S	CL28: CC28	82S181
27S29	DIP20: CL20	27S28
27S290	DIP24\600: DIP24	27S190
27S290L	CLR32	27S190
27S290S	CL28	27S190
27S291	DIP24\600: DIP24	27S190
27S291L	CLR32	27S190

Part Type	PADS 2000 Decal	PADS Logic Symbol
27S291S	CL28	27S190
27S31	DIP24\600	27S31
27S31L	CLR32	27S31L
27S31S	CL28	27S31L
27S32	DIP18	27S32
27S32L	CL20	27S32
27S33	DIP18	27S32
27S33L	CL20	27S32
27S35	DIP24	27S35
27S35L	CLR32	27S35
27S35S	CL28	27S35
27S37	DIP24	27S35
27S37L	CLR32	27S35
27S37S	CL28	27S35
27S41	DIP20	27S41
27S43	DIP24\600	RE4K8
27S43L	CLR32	RE4K8
27S43S	CL28	RE4K8
27S45	DIP24\600	RE4K8
27S45L	CLR32	RE4K8
27S45S	CL28	RE4K8
27S47	DIP24\600	RE4K8
27S47L	CLR32	RE4K8
27S47S	CL28	RE4K8

Part Type	PADS 2000 Decal	PADS Logic Symbol
27S49	DIP24\600	27S49
27S49L	CLR32	27S49
27S49S	CL28	27S49
27S51	DIP28\600	27S51
27S55	DIP24	27S55
27S65	DIP24	27S65
27S65L	CL28	27S65
27S75	DIP24	27S75
27S75L	CL28	27S75
27S85	DIP24	27S85
27S85L	CL28	27S85
27S95	DIP28\400	27S95
2817A	DIP28\600	RS2K8
2817A-LCC	CLR32	RE2K8
2817AL	CLR32	RS2K8
2864	DIP28\600	RS8K8: RS8K8-A
2864A	DIP28\600	RS8K8: RS8K8-A
2864AL	CLR32	RS8K8: RS8K8-A
2864B	DIP28\600	RS8K8: RS8K8-A
2864BL	CLR32	RS8K8: RS8K8-A
2864L	CLR32	RS8K8: RS8K8-A
28C010	DIP32\600	RS128K8: RS128K8-A
28C256	DIP28\600	RS32K8
28C256L	CLR32	RS32K8

Part Type	PADS 2000 Decal	PADS Logic Symbol
29PL141	DIP28\600: CL28	DIP26
2N2222	TO-18	NPN-V: NPN-VA: NPN-H: NPN-HA
2N2907	TO-18	PNP-H: PNP-V
2N2955	TO-3	PNP-V: PNP-VA: PNP-H: PNP-HA
2N3053	TO-39	NPN-V: NPN-VA: NPN-H: NPN-HA
2N3055	TO-3	NPN-V: NPN-VA: NPN-H: NPN-HA
2N4030	TO-39	NPN-V: NPN-VA: NPN-H: NPN-HA
31L01	DIP16	31L01
31L01L	CL20	31L01
32016	DIP48\600	32016
32032	CC68: PGA\68PB	32032
32081	DIP24\600	32081
32082	DIP48\600	32082
32201	DIP24\600	32201
32202	DIP40\600	32202
32440	DIP24\600	32440
32440V	CC28: CC28A	32440
32441	DIP24\600	32441
32441V	DIP24\600	32441
32442	DIP24\600	32442
32443	DIP24\600	32443
32490	DIP48\600	32490
32491	DIP24\600	32491
32492	DIP16	32492

Part Type	PADS 2000 Decal	PADS Logic Symbol
3440A	DIP16	3440
3441A	DIP16	3440
3443A	DIP16	3440
3446A	DIP16	3446
3447	DIP24:DIP24\600	3447
3448A	DIP16	3448A
3870	DIP40\600	3870
4000B	DIP14:DIP14\SO	NOR3 INV
4000C	DIP14:DIP14\SO	NOR3 INV
4000M	DIP14:DIP14\SO	NOR3 INV
4001	DIP14	NOR2
4001B	DIP14	NOR2
4001BC	DIP14	NOR2
4001BCP	DIP14	NOR2
4001BM	DIP14	NOR2
4001C	DIP14	NOR2
4001M	DIP14	NOR2
4002	DIP14	NOR4
4002B	DIP14	NOR4
4002BC	DIP14	NOR41
4002BM	DIP14	NOR4
4002C	DIP14	NOR4
4002M	DIP14	NOR4
4006	DIP14	4006

Part Type	PADS 2000 Decal	PADS Logic Symbol
4006B	DIP14	4006
4006BC	DIP14	4006
4006BM	DIP14	4006
4007	DIP14	4007
4007C	DIP14	4007
4007M	DIP14	4007
4007UB	DIP14	4007
4008	DIP16	4008
4008B	DIP16	4008
4008BC	DIP16	4008
4008BM	DIP16	4008
4009	DIP16	INV: INVDM
4009B	DIP16	INV: INVDM
4009C	DIP16	INV: INVDM
4009M	DIP16	INV: INVDM
4010	DIP16	INV: INVDM
40106	DIP14	INVST
40106B	DIP14: DIP14\SO	INVST
40106BC	DIP14	INVST
40106BM	DIP14	INVST
4010B	DIP16	INV: INVDM
4010C	DIP16	INV: INVDM
4010M	DIP16	INV: INVDM
4011	DIP14	NAND2

Part Type	PADS 2000 Decal	PADS Logic Symbol
4011B	DIP14:DIP14\SO	NAND2
4011BC	DIP14	NAND2
4011BM	DIP14	NAND2
4011C	DIP14	NAND2
4011M	DIP14	NAND2
4011UB	DIP14	NAND2
4012	DIP14	NAND4
4012B	DIP14	NAND4
4012BC	DIP14	NAND4
4012BM	DIP14	NAND4
4012C	DIP14	NAND4
4012M	DIP14	NAND4
4013	DIP14	\74
4013B	DIP14	\74
4013BC	DIP14	\74
4013BM	DIP14	\74
4014	DIP16	4014
4014B	DIP16	4014
4014BC	DIP16	4014
4014BM	DIP16	4014
4015	DIP16	4015
4015B	DIP16	4015
4015BC	DIP16	4015
4015BM	DIP16	4015

Part Type	PADS 2000 Decal	PADS Logic Symbol
4016	DIP14	4016
40160B	DIP16: DIP16\SO	\160
40160BC	DIP16: DIP16\SO	\160
40160BM	DIP16: DIP16\SO	\160
40161B	DIP16: DIP16\SO	\160
40161BC	DIP16: DIP16\SO	\160
40161BM	DIP16: DIP16\SO	\160
40162B	DIP16: DIP16\SO	\160
40162BC	DIP16: DIP16\SO	\160
40162BM	DIP16: DIP16\SO	\160
40163B	DIP16: DIP16\SO	\160
40163BC	DIP16: DIP16\SO	\160
40163BM	DIP16: DIP16\SO	\160
4016B	DIP14	4016
4016BC	DIP14	4016
4016BM	DIP14	4016
4017	DIP16	4017
40174B	DIP16: DIP16\SO	\174
40174BC	DIP16: DIP16\SO	\174
40174BM	DIP16: DIP16\SO	\174
40175B	DIP16: DIP16\SO	40175
40175BC	DIP16: DIP16\SO	40175
40175BM	DIP16: DIP16\SO	40175
4017B	DIP16	4017

Part Type	PADS 2000 Decal	PADS Logic Symbol
4017BC	DIP16	4017
4017BM	DIP16	4017
4018	DIP16	4018
4018B	DIP16	4018
4018BC	DIP16	4018
4018BM	DIP16	4018
4019	DIP16	4019
40192B	DIP16; DIP16\SO	\192
40192BC	DIP16; DIP16\SO	\192
40192BM	DIP16; DIP16\SO	\192
40193B	DIP16; DIP16\SO	40193
40193BC	DIP16; DIP16\SO	40193
40193BM	DIP16; DIP16\SO	40193
4019B	DIP16	4019
4019BC	DIP16	4019
4019BM	DIP16	4019
4020	DIP16	4020
4020B	DIP16	4020
4020BC	DIP16	4020
4020BM	DIP16	4020
4021	DIP16	4021
4021B	DIP16	4021
4021BC	DIP16	4021
4021BM	DIP16	4021

Part Type	PADS 2000 Decal	PADS Logic Symbol
4022	DIP16	4022
4022B	DIP16	4022
4022BC	DIP16	4022
4022BM	DIP16	4022
4023	DIP14	NAND3
4023B	DIP14	NAND3
4023BC	DIP14	NAND3
4023BCP	DIP14	NAND3
4023BM	DIP14	NAND3
4023C	DIP14	NAND3
4023M	DIP14	NAND3
4024	DIP14	4024
4024B	DIP14	4024
4024BC	DIP14	4024
4024BM	DIP14	4024
4025	DIP14	NOR3
4025B	DIP14	NOR3
4025BC	DIP14	NOR3
4025BM	DIP14	NOR3
4025C	DIP14	NOR3
4025M	DIP14	NOR3
4027	DIP16	4027
4027B	DIP16	4027
4027BC	DIP16	4027

Part Type	PADS 2000 Decal	PADS Logic Symbol
4027BM	DIP16	4027
4028	DIP16	4028
4028B	DIP16	4028
4028BC	DIP16	4028
4028BM	DIP16	4028
4029	DIP16	4029
4029B	DIP16	4029
4029BC	DIP16	4029
4029BM	DIP16	4029
4030	DIP14	XOR
4030B	DIP14	XOR
4030C	DIP14	XOR
4030M	DIP14	XOR
4031	DIP16	4031
4031B	DIP16	4031
4031BC	DIP16	4031
4031BM	DIP16	4031
4034	DIP24\600	4034
4034B	DIP24\600	4034
4034BC	DIP24\600	4034
4034BM	DIP24\600	4034
4035	DIP16	4035
4035B	DIP16	4035
4035BC	DIP16	4035

Part Type	PADS 2000 Decal	PADS Logic Symbol
4035BM	DIP16	4035
4040	DIP16	4020
4040B	DIP16	4020
4040BC	DIP16	4020
4040BM	DIP16	4020
4041	DIP14	4041
4041B	DIP14	4041
4041C	DIP14	4041
4041M	DIP14	4041
4042	DIP16	4042
4042B	DIP16	4042
4042BC	DIP16	4042
4042BM	DIP16	4042
4043	DIP16	4043
4043B	DIP16	4043
4043BC	DIP16	4043
4043BM	DIP16	4043
4044	DIP16: DIP16\SO	4044
4044B	DIP16: DIP16\SO	4044
4044BC	DIP16: DIP16\SO	4044
4044BM	DIP16: DIP16\SO	4044
4046	DIP16	4046
4046B	DIP16	4046
4046BC	DIP16	4046

Part Type	PADS 2000 Decal	PADS Logic Symbol
4046BM	DIP16	4046
4047	DIP14:DIP14\SO	4047
4047B	DIP14:DIP14\SO	4047
4047BC	DIP14:DIP14\SO	4047
4047BM	DIP14:DIP14\SO	4047
4048	DIP16:DIP16\SO	4048
4048B	DIP16:DIP16\SO	4048
4048BC	DIP16:DIP16\SO	4048
4048BM	DIP16:DIP16\SO	4048
4049	DIP16	INV:INVL:INVDM:INVDML
4049B	DIP16	INV:INVL:INVDM:INVDML
4049UB	DIP16	INV:INVL:INVDM:INVDML
4049UBC	DIP16	INV:INVL:INVDM:INVDML
4049UBM	DIP16	INV:INVL:INVDM:INVDML
405	DIP48\600	405
4050	DIP16	INVNON:INVNONL
4050B	DIP16	INVNON:INVNONL
4050BC	DIP16	INVNON:INVNONL
4050BCP	DIP16	INVNON:INVNONL
4050BM	DIP16	INVNON:INVNONL
4050M	DIP16	INVNON:INVNONL
4051B	DIP16:DIP16\SO	4051
4051BC	DIP16:DIP16\SO	4051
4051BM	DIP16:DIP16\SO	4051

Part Type	PADS 2000 Decal	PADS Logic Symbol
4052	DIP16	4052
4052B	DIP16	4052
4052BC	DIP16	4052
4052BM	DIP16	4052
4053	DIP16	4052
4053B	DIP16	4052
4053BC	DIP16	4052
4053BM	DIP16	4052
4060	DIP16	4060
4060B	DIP16	4060
4060BC	DIP16	4060
4060BM	DIP16	4060
4061B	DIP16: DIP16\SO	4060
4066	DIP14	4016
4066B	DIP14	4016
4066BC	DIP14	4016
4066BM	DIP14	4016
4068B	DIP14: DIP14\SO	NAND8
4069	DIP14	INV: INVDM
4069B	DIP14	INV: INVDM
4069UB	DIP14	INV: INVDM
4069UBC	DIP14	INV: INVDM
4069UBM	DIP14	INV: INVDM
4070	DIP14	XOR

Part Type	PADS 2000 Decal	PADS Logic Symbol
4070B	DIP14	XOR
4070BC	DIP14	XOR
4070BM	DIP14	XOR
4071	DIP14	OR2
4071B	DIP14	OR2
4071BC	DIP14	OR2
4071BM	DIP14	OR2
4072	DIP14	OR4
4072B	DIP14	OR4
4072BC	DIP14	OR4
4072BM	DIP14	OR4
4073	DIP14	AND3
4073B	DIP14	AND3
4073BC	DIP14	AND3
4073BM	DIP14	AND3
4075	DIP14	OR3
4075B	DIP14	OR3
4075BC	DIP14	OR3
4075BM	DIP14	OR3
4076	DIP16	4076
4076B	DIP16	4076
4076BC	DIP16	4076
4076BM	DIP16	4076
4077	DIP14:DIP14\SO	XNOR

Part Type	PADS 2000 Decal	PADS Logic Symbol
4077B	DIP14: DIP14\SO	XNOR
4078B	DIP14: DIP14\SO	NOR8
4081	DIP14	AND2
4081B	DIP14	AND2
4081BC	DIP14	AND2
4081BM	DIP14	AND2
4082	DIP14	AND4
4082B	DIP14	AND4
4082BC	DIP14	AND4
4082BM	DIP14	AND4
4089	DIP16	4089
4089B	DIP16	4089
4089BC	DIP16	4089
4089BM	DIP16	4089
4093	DIP14	NAND2: NAND2DM
4093B	DIP14	NAND2: NAND2DM
4093BC	DIP14	NAND2: NAND2DM
4093BM	DIP14	NAND2: NAND2DM
4094	DIP16	4094
4094B	DIP16	4094
4094BC	DIP16	4094
4094BM	DIP16	4094
4099	DIP16	4099
4099B	DIP16	4099

Part Type	PADS 2000 Decal	PADS Logic Symbol
4099BC	DIP16	4099
41464	DIP18	RD64K4
4316B	DIP16: DIP16\SO	4316
4351B	DIP20: DIP20\SOL	4351
4352B	DIP20: DIP20\SOL	4352
4353B	DIP20: DIP20\SOL	4353
4511B	DIP16: DIP16\SO	4511
4512B	DIP16: DIP16\SO	4512
4518B	DIP16: DIP16\SO	\4518
4520B	DIP16: DIP16\SO	\4518
4527B	DIP16	4089
4528B	DIP16: DIP16\SO	\4538
4538B	DIP16: DIP16\SO	\4538
4539B	DIP16: DIP16\SO	\153
4555B	DIP16: DIP16\SO	4555
4556B	DIP16: DIP16\SO	4556
4724B	DIP16: DIP16\SO	4724
48416	DIP18	RD16K4
4864	DIP16	RD256K1
50256	DIP16	RD256K1
50257	DIP16	RD256K1
511000	DIP18	RD1024K1
514256	DIP20	RD256K4
6116	DIP24\600	6116

Part Type	PADS 2000 Decal	PADS Logic Symbol
6167	DIP20	6167
6167SA	DIP20	6167
6167SL	DIP20	6167
6168	DIP20: CLR20	6168
6208	DIP24	RS64K4
62256	DIP28\600: DIP28\SOL	RS32K8
6264	DIP28\600: DIP28\SOL	RE8K8
6287	DIP22	6287
63484	DIP64\900	63484
68000	DIP64\900	68000
68000PGA	PGA\68PA	68000
68000PLCC	CC68	68000
6801	DIP40\600	DIP38
68010	DIP64\900	68000
68010A	CC68	68000
68010PGA	PGA\68PA	68000
6801U4	DIP40\600	DIP38
6802	DIP40\600	6802
6802NS	DIP40\600	6802
6803	DIP40\600	DIP38
6803E	DIP40\600	DIP38
6803U4	DIP40\600	DIP38
6804J2	DIP20	DIP18
6804P2	DIP28\600	DIP26

Part Type	PADS 2000 Decal	PADS Logic Symbol
6805K2	DIP40\600	DIP40
6805K3	DIP40\600	DIP40
6805P2	DIP28\600	DIP26
6805P4	DIP28\600	DIP26
6805P6	DIP28\600	DIP26
6805R2	DIP40\600	DIP40
6805R3	DIP40\600	DIP38
6805S2	DIP28\600	DIP26
6805S3	DIP28\600	DIP26
6805T2	DIP28\600	6805
6805U2	DIP40\600	DIP38
6805U3	DIP40\600	DIP38
6808	DIP40\600	6802
6809	DIP40\600	DIP38
6809E	DIP40\600	DIP38
68120	DIP48\600	DIP46
68121	DIP48\600	DIP46
68154	DIP40\600	68154
68154-PLCC	CC44	68154
68154A	CC44	68154
68155	DIP40\600	68155
68155A	CC44	68155
68155PLCC	CC44	68155
68171	DIP16	68171

Part Type	PADS 2000 Decal	PADS Logic Symbol
68172	DIP28\600	68172
68172A	CC44	68172
68172PLCC	CC44	68172
68173	DIP28\600	68173
6821	DIP40\600	DIP38
6822	DIP40\600	DIP38
6829	DIP40\600	DIP38
6835	DIP40\600	DIP38
6839	DIP24\600	6839
6840	DIP28\600	6840
68430	DIP48\600	68430
68430A	CC52	68430
68430PLCC	CC52	68430
6844	DIP40\600	DIP38
6845	DIP40\600	6845
6846	DIP40\600	6846
6847	DIP40\600	6847
6847Y	DIP40\600	6847
6850	DIP24\600	DIP22
6852	DIP24\600	6852
6854	DIP28\600	6854
68562	DIP48\600	68562
68562A	DIP48\600	68562
6859	DIP24\600	DIP22

Part Type	PADS 2000 Decal	PADS Logic Symbol
68661	DIP28\600; CC28	2661
68681	DIP40\600	2681N40
68681-PLCC	CC44	2681N40
68681A	CC44; CC44A	2681N40
68692	DIP40\600	2681N40
68692-PLCC	CC44; CC44A	2681N40
68692A	CC44; CC44A	2681N40
68701	DIP40\600	DIP38
68701U4	DIP40\600	DIP38
68705P3	DIP28\600	DIP26
68705P5	DIP28\600	DIP26
68705R3	DIP40\600	DIP38
68705R5	DIP40\600	DIP38
68705U3	DIP40\600	DIP38
68705U5	DIP40\600	DIP38
6875	DIP16	6875
6880A	DIP16	6880A
6882A	DIP20	DIP20
6882B	DIP20	DIP20
6883	DIP40\600	6883
6885	DIP16	DIP16
6886	DIP16	DIP16
6887	DIP16	DIP16
6888	DIP16	DIP16

Part Type	PADS 2000 Decal	PADS Logic Symbol
6889	DIP16	DIP16
6890	DIP20	6890
68HC04J2	DIP20	DIP18
68HC04J3	DIP20	DIP18
68HC04P2	DIP28\600	68HC04
68HC04P3	DIP28\600	68HC04
68HC09E	DIP40\600	68HC09
68HC34	DIP40\600	DIP38
68HC51	DIP28\600	68HC51
68HC68A1	DIP16	68HC68
68HC68R1	DIP8	68HC68R1
68HC68R2	DIP8	68HC68R1
7990	DIP48\600	7990
7992A	DIP24\300	7992
80186	PGA\68P: CC68A	DIP64
80188	PGA\68P: CC68A	DIP64
80286	PGA\68P	80286
8031	DIP40\600	8031
8031AH	DIP40\600	8031
8032AH	DIP40\600	8031
8032BH	DIP40\600	8031
8035AHL	DIP40\600	8035AHL
8035H	DIP40\600	8035AHL
8039AHL	DIP40\600	8035AHL

Part Type	PADS 2000 Decal	PADS Logic Symbol
8039H	DIP40\600	8035AHL
8040AHL	DIP40\600	8035AHL
8041AH	DIP40\600	8041
8042AH	DIP40\600	8042
8044AH	DIP40\600	8031
8048AH	DIP40\600	8035AHL
8049AH	DIP40\600	8035AHL
8050AH	DIP40\600	8035AHL
8051	DIP40\600	8031
8051AH	DIP40\600	8031
8052	DIP40\600	8031
8052AH	DIP40\600	8031
8052BH	DIP40\600	8031
8080A	DIP40\600	8088
8085AH	DIP40\600	8085AH
8086	DIP40\600	8086
8088	DIP40\600	8088
8094	DIP48\600	8094
8095	DIP48\600	8094
8095BH	DIP48\600	8094
8095CH	DIP48\600	8094
8096	PGA\68P	8096
8096BH	PGA\68P	8096
8097	PGA\68P	8096

Part Type	PADS 2000 Decal	PADS Logic Symbol
8097BH	PGA\68P	8096
80C252	DIP40\600	80C252
80C31BH	DIP40\600	8031
80C32AH	DIP40\600	8031
80C39	DIP40\600	8035AHL
80C51BH	DIP40\600	80C252
80C52AH	DIP40\600	8031
8152A	DIP48\600	8152
8152AL	CL52	8152
8153A	DIP48\600	8152
8153AL	CL52	8152
8155H	DIP40\600	8155H
8156H	DIP40\600	8155H
8185	DIP18	8185
8203	DIP40\600	8203
8206	PGA\68P	8206
82064	DIP40\600	82064
8207	PGA\68P	8207
82072	DIP40\600	82072
82188	DIP28\600	82188
8224	DIP16	8224
8228	DIP28\600	8228
82284	DIP18	DIP16
82288	DIP20	DIP18

Part Type	PADS 2000 Decal	PADS Logic Symbol
8231A	DIP24\600	8231
8237A	DIP40\600	8237A
8238	DIP28\600	8228
8243	DIP24\600	8243
8250A	DIP40\600	8250
8251A	DIP28\600	8251A
8253	DIP24\600	8253
8254	DIP24\600	8253
8255A	DIP40\600	8255A
8256AH	DIP40\600	8256AH
8257	DIP40\600	8257
8259A	DIP28\600	8259A
82716	PGA\68P: CC68A	82716
8272A	DIP40\600	8272A
82730	PGA\68P	82730
8275H	DIP40\600	8275
8276H	DIP40\600	8276
82786	PGA\88P	82786
8279	DIP40\600	8279
8284A	DIP18	8284
8288	DIP20	8288
8289	DIP20	8289
82C08	DIP48\600	82C08
82C55A	DIP40\600	8255A

Part Type	PADS 2000 Decal	PADS Logic Symbol
82C59A	DIP28\600	DIP26
82HS195	DIP20	82HS195
82HS321	DIP24\600	82HS321
82HS641	DIP24\600	82HS641
82LS16	DIP16	82LS16
82S115	DIP24\600	82S115
82S123	DIP16	82S123
82S126	DIP16	82S126
82S129	DIP16	82S129
82S130	DIP16	82S130
82S131	DIP16	82S130
82S181	DIP24\600	82S181
82S185	DIP18	82S185
82S191	DIP24\600	82S191
82S191CN3	DIP24	82S191
82S23	DIP16	82S123
82S25	DIP16	82S25
8340	DIP24\600	8340
8341	DIP24\600	8341
8342	DIP24\600	8342
8343	DIP24\600	8343
8344AH	DIP40\600	8031
8394	DIP48\600	8094
8395	DIP48\600	8094

Part Type	PADS 2000 Decal	PADS Logic Symbol
8395BH	DIP48\600	8094
8396	PGA\68P	8096
8396BH	PGA\68P	8096
8397	PGA\68P	8096
8397BH	PGA\68P	8096
83C154	DIP40\600	DIP38
83C452P	PGA\68P	DIP64
8741AH	DIP40\600	8041
8742AH	DIP40\600	DIP38
8744H	DIP40\600	8031
8748H	DIP40\600	8035AHL
8749H	DIP40\600	8035AHL
8751BH	DIP40\600	8031
8751H	DIP40\600	8031
8752BH	DIP40\600	8031
8755A	DIP40\600	DIP38
8795BH	DIP48\600	DIP44
87C256	DIP28\600	RE32K8
87C452P	PGA\68P	DIP64
87C51	DIP40\600	8031
87C51-PLCC	CC44: CC44A	8031
87C51FA	DIP40\600	DIP38
87C64	DIP28\600	RE8K8
87C652	DIP40\600	DIP38

Part Type	PADS 2000 Decal	PADS Logic Symbol
8X305	DIP50\900	8X305
8X310	DIP40\600	8X310
8X320	DIP40\600	8X320
8X372	DIP24\400	8X372
8X376	DIP24\400	8X372
9044	DIP18	9044
90C644	DIP24\400	90C644
9114	DIP18	9114
9124	DIP18	9114
9128	DIP24\600	9128
9151	DIP24	9151
93L412	DIP22\400	93L412
93L412L	CLR28	93L412
93L415	DIP16	93L415
93L415L	CLR20	93L415
93L422	DIP22\400	93L412
93L422L	CLR28	93L412
93L425	DIP16	93L415
93L425L	CLR20	93L415
93L469	DIP24	93L469
9864	DIP28\600	9864
9864L	CLR32	9864
99C10	DIP24\400	99C10
99C164	DIP22	99C164

Part Type	PADS 2000 Decal	PADS Logic Symbol
99C19	DIP28\600	99C19
99C328	DIP28\600	99C328
99C416	DIP40\600	99C416
99C58	DIP24	99C58
99C58L	CLR28	99C58
99C59	DIP24	99C58
99C59L	CLR28	99C58
99C60	DIP24	99C60
99C60L	CLR28	99C60
99C641	DIP22: CLR22	99C641
99C89	DIP28\600	99C89
99CL68	DIP20	99CL68
99CL88	DIP28\600	99C88
99CL88L	CLR32	99C88
A8096BH	PGA\68P	DIP64
A8097BH	PGA\68P	DIP64
A8796BH	PGA\68P	DIP64
A8797BH	PGA\68P	DIP64
ACOMPD	DIP8: DIP8\SO	AOPAMP: AOPAMPM
ACOMPQ	DIP14: DIP14\SO	AOPAMP: AOPAMPM
ADARL-8	DIP18	ADARL-8
AM29818	DIP24	\29818
AM29818-CC	CC28	\29818
AM29821	DIP24	\29821

Part Type	PADS 2000 Decal	PADS Logic Symbol
AM29821-CC	CC28	\29821
AM29822	DIP24	\29822
AM29822-CC	CC28	\29822
AM29823	DIP24	\29823
AM29823-CC	CC28	\29823
AM29824	DIP24	\29824
AM29824-CC	CC28	\29824
AM29825	DIP24	\29825
AM29825-CC	CC28	\29825
AM29826	DIP24	\29826
AM29826-CC	CC28	\29826
AM29827	DIP24	\29827
AM29828	DIP24	\29828
AM29861	DIP24	\29861
AM29862	DIP24	\29862
AM29863	DIP24	\29863
AM29864	DIP24	\29864
AOPAMPD	DIP8: DIP8\SO	AOPAMP: AOPAMPM
AOPAMPQ	DIP14: DIP14\SO	AOPAMP: AOPAMPM
BAW62	DO7: DO35: DO41	DIO-H: DIO-V
BC548	TO-92	NPN-V: NPN-VA: NPN-H: NPN-HA
BC558	TO-92	PNP-V: PNP-VA: PNP-H: PNP-HA
BC639	TO-92	NPN-V: NPN-VA: NPN-H: NPN-HA
BC640	TO-92	PNP-V: PNP-VA: PNP-H: PNP-HA

Part Type	PADS 2000 Decal	PADS Logic Symbol
BF199	TO-92	NPN-V: NPN-VA: NPN-H: NPN-HA
BF245	TO-92	JFETN-V
BF926	TO-92	PNP-V: PNP-VA: PNP-H: PNP-HA
BSJ174	TO-92	JFETP-V
BST100	TO-92	MFETP-V
BST72	TO-92	MFETN-V
BT139	TO-220	TRIAC-V
BT149-E	TO-92	SCR-V
BT149-M	TO-92	SCR-V
BT152	TO-220	SCR-V
BUZ71	TO-220	MFETN-V
BZT03	DO7: DO35: DO41	DIOZ-H: DIOZ-V
BZV85	DO7: DO35: DO41	DIOZ-H: DIOZ-V
BZX79	DO7: DO35: DO41	DIOZ-H: DIOZ-V
CAP	DCAP\SR21	CAP: CAPA: CAPH: CAPAH
CAP1	CAP1	CAP: CAPA: CAPH: CAPAH
CAP1\5G	CAP1	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CAPSMD	CC1206: CC1210: CC1812	CAP: CAPA: CAPH: CAPAH
CAPSMD	CC1206: CC1210: CC1812	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CAP\5G	DCAP\SR21	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CAP\CR08	CAP\CR08	CAP: CAPA: CAPH: CAPAH
CAP\CR08\5G	CAP\CR08	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CAP\MA10	CAP\MA10	CAP: CAPA: CAPH: CAPAH
CAP\MA10\5G	CAP\MA10	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G

Part Type	PADS 2000 Decal	PADS Logic Symbol
CAP\MA20	CAP\MA20	CAP: CAPA: CAPH: CAPAH
CAP\MA20\5G	CAP\MA20	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CAP\MA30	CAP\MA30	CAP: CAPA: CAPH: CAPAH
CAP\MA30\5G	CAP\MA30	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CAP\MA40	CAP\MA40	CAP: CAPA: CAPH: CAPAH
CAP\MA40\5G	CAP\MA40	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CAP\MA50	CAP\MA50	CAP: CAPA: CAPH: CAPAH
CAP\MA50\5G	CAP\MA50	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CAP\MA60	CAP\MA60	CAP: CAPA: CAPH: CAPAH
CAP\MA60\5G	CAP\MA60	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CAP\MR04	CAP\MR04	CAP: CAPA: CAPH: CAPAH
CAP\MR04\5G	CAP\MR04	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CAP\MR05	CAP\MR05	CAP: CAPA: CAPH: CAPAH
CAP\MR05\5G	CAP\MR05	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CAP\MR06	CAP\MR06	CAP: CAPA: CAPH: CAPAH
CAP\MR06\5G	CAP\MR06	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CAP\MR07	CAP\MR07	CAP: CAPA: CAPH: CAPAH
CAP\MR07\5G	CAP\MR07	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CAP\MR08	CAP\MR08	CAP: CAPA: CAPH: CAPAH
CAP\MR08\5G	CAP\MR08	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CC05	CAP\MR05	CAP+: CAPA+: CAPH+: CAPAH+
CC05\5G	CAP\MR05	CAP\5G+: CAPA\5G+: CAPH\5G+: CAPAH\5G+
CC06	CAP\MR06	CAP+: CAPA+: CAPH+: CAPAH+
CC06\5G	CAP\MR06	CAP\5G+: CAPA\5G+: CAPH\5G+: CAPAH\5G+

Part Type	PADS 2000 Decal	PADS Logic Symbol
CC07	CAP\MR07	CAP+: CAPA+: CAPH+: CAPAH+
CC07\5G	CAP\MR07	CAP\5G+: CAPA\5G+: CAPH\5G+: CAPAH\5G+
CC08	CAP\MR08	CAP+: CAPA+: CAPH+: CAPAH+
CC08\5G	CAP\MR08	CAP\5G+: CAPA\5G+: CAPH\5G+: CAPAH\5G+
CC09	CAP\MR04	CAP+: CAPA+: CAPH+: CAPAH+
CC09\5G	CAP\MR04	CAP\5G+: CAPA\5G+: CAPH\5G+: CAPAH\5G+
CC20	CC20	GEN\DIP20
CC20A	CC20A	GEN\DIP20
CC28	CC28	GEN\DIP28
CC28A	CC28A	GEN\DIP28
CC40\40	CC40\40	GEN\DIP40
CC44	CC44	GEN\DIP44
CC44A	CC44A	GEN\DIP44
CC52	CC52	GEN\DIP52
CC68	CC68	GEN\DIP68
CC68A	CC68A	GEN\DIP68
CC84	CC84	GEN\DIP84
CC84A	CC84A	GEN\DIP84
CCR05	CAP\MR05	CAP+: CAPA+: CAPH+: CAPAH+
CCR05\5G	CAP\MR05	CAP\5G+: CAPA\5G+: CAPH\5G+: CAPAH\5G+
CCR06	CAP\MR06	CAP+: CAPA+: CAPH+: CAPAH+
CCR06\5G	CAP\MR06	CAP\5G+: CAPA\5G+: CAPH\5G+: CAPAH\5G+
CCR07	CAP\MR07	CAP+: CAPA+: CAPH+: CAPAH+
CCR07\5G	CAP\MR07	CAP\5G+: CAPA\5G+: CAPH\5G+: CAPAH\5G+

Part Type	PADS 2000 Decal	PADS Logic Symbol
CCR08	CAP\MR08	CAP+: CAPA+: CAPH+: CAPAH+
CCR08\5G	CAP\MR08	CAP\5G+: CAPA\5G+: CAPH\5G+: CAPAH\5G+
CCR09	CAP\MR04	CAP+: CAPA+: CAPH+: CAPAH+
CCR09\5G	CAP\MR04	CAP\5G+: CAPA\5G+: CAPH\5G+: CAPAH\5G+
CCR18	CCR18	GEN\DIP18
CCR32	CCR32	GEN\DIP32
CCR75	CAP\MA10	CAP+: CAPA+: CAPH+: CAPAH+
CCR75\5G	CAP\MA10	CAP\5G+. CAPA\5G+: CAPH\5G+: CAPAH\5G+
CCR76	CAP\MA20	CAP+: CAPA+: CAPH+: CAPAH+
CCR76\5G	CAP\MA20	CAP\5G+: CAPA\5G+: CAPH\5G+: CAPAH\5G+
CCR77	CAP\MA40	CAP+: CAPA+: CAPH+: CAPAH+
CCR77\5G	CAP\MA40	CAP\5G+: CAPA\5G+: CAPH\5G+: CAPAH\5G+
CCR78	CAP\MA50	CAP+: CAPA+: CAPH+: CAPAH+
CCR78\5G	CAP\MA50	CAP\5G+: CAPA\5G+: CAPH\5G+: CAPAH\5G+
CCR79	CAP\MA60	CAP+: CAPA+: CAPH+: CAPAH+
CCR79\5G	CAP\MA60	CAP\5G+: CAPA\5G+: CAPH\5G+: CAPAH\5G+
CD4015	DIP16: DIP16\SO	4015
CD4017	DIP16: DIP16\SO	4017
CD4020	DIP16: DIP16\SO	4040
CD4022	DIP16: DIP16\SO	\4022
CD4024	DIP14: DIP14\SO	\4024
CD4046	DIP16: DIP16\SO	\4046
CD4049	DIP16: DIP16\SO	INV: INVL: INVDM: INVDML
CD4051	DIP16: DIP16\SO	4051

Part Type	PADS 2000 Decal	PADS Logic Symbol
CD4052	DIP16:DIP16\SO	4052
CD4053	DIP16:DIP16\SO	\4053
CD4060	DIP16:DIP16\SO	\4060
CD4066	DIP14:DIP14\SO	4066:4066V
CD4075	DIP14:DIP14\SO	OR3
CD4094	DIP16:DIP16\SO	4094
CK05	CAP\MR05	CAP:CAPA:CAPH:CAPAH
CK05\5G	CAP\MR05	CAP\5G:CAPA\5G:CAPH\5G:CAPAH\5G
CK06	CAP\MR06	CAP:CAPA:CAPH:CAPAH
CK06\5G	CAP\MR06	CAP\5G:CAPA\5G:CAPH\5G:CAPAH\5G
CK12	CAP\MA10	CAP:CAPA:CAPH:CAPAH
CK12\5G	CAP\MA10	CAP\5G:CAPA\5G:CAPH\5G:CAPAH\5G
CK13	CAP\MA20	CAP:CAPA:CAPH:CAPAH
CK13\5G	CAP\MA20	CAP\5G:CAPA\5G:CAPH\5G:CAPAH\5G
CK14	CAP\MA40	CAP:CAPA:CAPH:CAPAH
CK14\5G	CAP\MA40	CAP\5G:CAPA\5G:CAPH\5G:CAPAH\5G
CK15	CAP\MA50	CAP:CAPA:CAPH:CAPAH
CK15\5G	CAP\MA50	CAP\5G:CAPA\5G:CAPH\5G:CAPAH\5G
CK16	CAP\MA60	CAP:CAPA:CAPH:CAPAH
CK16\5G	CAP\MA60	CAP\5G:CAPA\5G:CAPH\5G:CAPAH\5G
CKR04	CAP\MR04	CAP:CAPA:CAPH:CAPAH
CKR04\5G	CAP\MR04	CAP\5G:CAPA\5G:CAPH\5G:CAPAH\5G
CKR05	CAP\MR05	CAP:CAPA:CAPH:CAPAH
CKR05\5G	CAP\MR05	CAP\5G:CAPA\5G:CAPH\5G:CAPAH\5G

Part Type	PADS 2000 Decal	PADS Logic Symbol
CKR06	CAP\MR06	CAP: CAPA: CAPH: CAPAH
CKR06\5G	CAP\MR06	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CKR08	CAP\CR08	CAP: CAPA: CAPH: CAPAH
CKR08\5G	CAP\CR08	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CKR11	CAP\MA10	CAP: CAPA: CAPH: CAPAH
CKR11\5G	CAP\MA10	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CKR12	CAP\MA20	CAP: CAPA: CAPH: CAPAH
CKR12\5G	CAP\MA20	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CKR14	CAP\MA40	CAP: CAPA: CAPH: CAPAH
CKR14\5G	CAP\MA40	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CKR15	CAP\MA50	CAP: CAPA: CAPH: CAPAH
CKR15\5G	CAP\MA50	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CKR16	CAP\MA60	CAP: CAPA: CAPH: CAPAH
CKR16\5G	CAP\MA60	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CL20	CL20	GEN\DIP20
CL25	CL25	CAP: CAPA: CAPH: CAPAH
CL28	CL28	GEN\DIP28
CL44	CL44	GEN\DIP44
CL52	CL52	GEN\DIP52
CLR18	CLR18	GEN\DIP18
CLR20	CLR20	GEN\DIP20
CLR28	CLR28	GEN\DIP28
CLR32	CLR32	GEN\DIP32
CON\SIP\1P	SIP\1P	EXTIN: EXTOUT

Part Type	PADS 2000 Decal	PADS Logic Symbol
CON\SIP\10P	SIP\10P	EXTIN: EXTOUT
CON\SIP\11P	SIP\11P	EXTIN: EXTOUT
CON\SIP\12P	SIP\12P	EXTIN: EXTOUT
CON\SIP\13P	SIP\13P	EXTIN: EXTOUT
CON\SIP\14P	SIP\14P	EXTIN: EXTOUT
CON\SIP\15P	SIP\15P	EXTIN: EXTOUT
CON\SIP\16P	SIP\16P	EXTIN: EXTOUT
CON\SIP\17P	SIP\17P	EXTIN: EXTOUT
CON\SIP\18P	SIP\18P	EXTIN: EXTOUT
CON\SIP\19P	SIP\19P	EXTIN: EXTOUT
CON\SIP\2P	SIP\2P	EXTIN: EXTOUT
CON\SIP\20P	SIP\20P	EXTIN: EXTOUT
CON\SIP\3P	SIP\3P	EXTIN: EXTOUT
CON\SIP\4P	SIP\4P	EXTIN: EXTOUT
CON\SIP\5P	SIP\5P	EXTIN: EXTOUT
CON\SIP\6P	SIP\6P	EXTIN: EXTOUT
CON\SIP\7P	SIP\7P	EXTIN: EXTOUT
CON\SIP\8P	SIP\8P	EXTIN: EXTOUT
CON\SIP\9P	SIP\9P	EXTIN: EXTOUT
C\0805	CC0805	CAP: CAPA: CAPH: CAPAH
C\0805\5G	CC0805	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
C\1206	CC1206	CAP: CAPA: CAPH: CAPAH
C\1206\5G	CC1206	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
C\1210	CC1210	CAP: CAPA: CAPH: CAPAH

Part Type	PADS 2000 Decal	PADS Logic Symbol
C\1210\5G	CC1210	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
C\1812	CC1812	CAP: CAPA: CAPH: CAPAH
C\1812\5G	CC1812	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
DCAP1	DCAP1	CAP: CAPA: CAPH: CAPAH
DCAP1\5G	DCAP1	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
DCAP\SR15	DCAP\SR15	CAP: CAPA: CAPH: CAPAH
DCAP\SR15\5G	DCAP\SR15	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
DCAP\SR20	DCAP\SR20	CAP: CAPA: CAPH: CAPAH
DCAP\SR20\5G	DCAP\SR20	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
DCAP\SR21	DCAP\SR21	CAP: CAPA: CAPH: CAPAH
DCAP\SR21\5G	DCAP\SR21	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
DCAP\SR26	DCAP\SR26	CAP: CAPA: CAPH: CAPAH
DCAP\SR26\5G	DCAP\SR26	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
DCAP\SR27	DCAP\SR27	CAP: CAPA: CAPH: CAPAH
DCAP\SR27\5G	DCAP\SR27	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
DCON25F	DCON25F	EXTIN: EXTOUT
DCON25M	DCON25M	EXTIN: EXTOUT
DCON9F	DCON9F	EXTIN: EXTOUT
DCON9M	DCON9M	EXTIN: EXTOUT
DIODE	DO7: DO35: DO41	DIO-H: DIO-V
DIODEZ	DO7: DO35: DO41	DIOZ-H: DIOZ-V
DIP10	DIP10	DIP10
DIP14	DIP14	GEN\DIP14
DIP14\SO	DIP14\SO	GEN\DIP14

Part Type	PADS 2000 Decal	PADS Logic Symbol
DIP16	DIP16	GEN\DIP16
DIP16\SO	DIP16\SO	GEN\DIP16
DIP16\SOL	DIP16\SOL	GEN\DIP16
DIP18	DIP18	GEN\DIP18
DIP20	DIP20	GEN\DIP20
DIP20\SOL	DIP20\SOL	GEN\DIP20
DIP22	DIP22	GEN\DIP22
DIP22\300	DIP22\300	GEN\DIP22
DIP22\400	DIP22\400	GEN\DIP22
DIP24	DIP24	GEN\DIP24
DIP24\300	DIP24\300	GEN\DIP24
DIP24\400	DIP24\400	GEN\DIP24
DIP24\600	DIP24\600	GEN\DIP24
DIP24\SOL	DIP24\SOL	GEN\DIP24
DIP28\600	DIP28\600	GEN\DIP28
DIP28\SOL	DIP28\SOL	GEN\DIP28
DIP32\600	DIP32\600	GEN\DIP32
DIP4(14)	DIP4(14)	GEN\DIP4
DIP40\600	DIP40\600	GEN\DIP40
DIP48\600	DIP48\600	GEN\DIP48
DIP50\900	DIP50\900	GEN\DIP50
DIP52\600	DIP52\600	GEN\DIP52
DIP52\900	DIP52\900	GEN\DIP52
DIP6	DIP6	GEN\DIP6

Part Type	PADS 2000 Decal	PADS Logic Symbol
DIP64\900	DIP64\900	GEN\DIP64
DIP8	DIP8	GEN\DIP8
DIP8\300	DIP8\300	GEN\DIP8
DIP8\SO	DIP8\SO	GEN\DIP8
DO35	DO35	DIO-H: DIO-V
DO41	DO41	DIO-H: DIO-V
DO7	DO7	DIO-H: DIO-V
GAL20V8	DIP24	PAL20V8
HEADER10	HEADER10	EXTIN: EXTOUT
HEADER20	HEADER20	EXTIN: EXTOUT
HEADER34	HEADER34	EXTIN: EXTOUT
ICM7555	DIP8: DIP8\SO	NE555
ICM7555-SO	DIP8\SO	NE555
LM324	DIP14: DIP14\SO	AOPAMP: AOPAMPM AOPAMPPN
LM339	DIP14: DIP14\SO	AOPAMP: AOPAMPM AOPAMPPN
LM358	DIP8: DIP8\SO	AOPAMP: AOPAMPM AOPAMPPN
LM3900	DIP14: DIP14\SO	AOPAMP: AOPAMPM AOPAMPPN
LM741	DIP8: DIP8\SO	741
MA10	CAP\MA10	CAP: CAPA: CAPH: CAPAH
MA10\5G	CAP\MA10	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
MA20	CAP\MA20	CAP: CAPA: CAPH: CAPAH
MA20\5G	CAP\MA20	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
MA30	CAP\MA30	CAP: CAPA: CAPH: CAPAH
MA30\5G	CAP\MA30	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G

Part Type	PADS 2000 Decal	PADS Logic Symbol
MA40	CAP\MA40	CAP: CAPA: CAPH: CAPAH
MA40\5G	CAP\MA40	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
MA50	CAP\MA50	CAP: CAPA: CAPH: CAPAH
MA50\5G	CAP\MA50	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
MA60	CAP\MA60	CAP: CAPA: CAPH: CAPAH
MA60\5G	CAP\MA60	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
MC10	CAP\MA10	CAP: CAPA: CAPH: CAPAH
MC10\5G	CAP\MA10	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
MC20	CAP\MA40	CAP: CAPA: CAPH: CAPAH
MC20\5G	CAP\MA40	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
MC50	CAP\MR05	CAP: CAPA: CAPH: CAPAH
MC50\5G	CAP\MR05	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
MC54	CAP\MR04	CAP: CAPA: CAPH: CAPAH
MC54\5G	CAP\MR04	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
MC60	CAP\MR06	CAP: CAPA: CAPH: CAPAH
MC60\5G	CAP\MR06	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
MC70	CAP\MA20	CAP: CAPA: CAPH: CAPAH
MC70\5G	CAP\MA20	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
MC78	CAP\MA30	CAP: CAPA: CAPH: CAPAH
MC78\5G	CAP\MA30	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
MC85	CAP\MA50	CAP: CAPA: CAPH: CAPAH
MC85\5G	CAP\MA50	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
MC87	CAP\MA60	CAP: CAPA: CAPH: CAPAH
MC87\5G	CAP\MA60	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G

Part Type	PADS 2000 Decal	PADS Logic Symbol
MC95	CAP\MR07	CAP: CAPA: CAPH: CAPAH
MC95\5G	CAP\MR07	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
MC96	CAP\MR08	CAP: CAPA: CAPH: CAPAH
MC96\5G	CAP\MR08	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
MOIRE	MOIRE	
MR04	CAP\MR04	CAP: CAPA: CAPH: CAPAH
MR04\5G	CAP\MR04	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
MR05	CAP\MR05	CAP: CAPA: CAPH: CAPAH
MR05\5G	CAP\MR05	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
MR06	CAP\MR06	CAP: CAPA: CAPH: CAPAH
MR06\5G	CAP\MR06	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
MR07	CAP\MR07	CAP: CAPA: CAPH: CAPAH
MR07\5G	CAP\MR07	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
MR08	CAP\MR08	CAP: CAPA: CAPH: CAPAH
MR08\5G	CAP\MR08	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
MTHOLE1	MTHOLE1	
MTHOLE2	MTHOLE2	
N8031	CC44: CC44A	8031
N8031AH	CC44: CC44A	8031
N8035AHL	CC44: CC44A	8035AHL
N8035H	CC44: CC44A	8035AHL
N8039AHL	CC44: CC44A	8035AHL
N8039H	CC44: CC44A	8035AHL
N8040AHL	CC44: CC44A	8035AHL

Part Type	PADS 2000 Decal	PADS Logic Symbol
N8044AH	CC44: CC44A	8031
N8048AH	CC44: CC44A	8035AHL
N8048AHL	CC44: CC44A	8035AHL
N8049AH	CC44: CC44A	8035AHL
N8050AH	CC44: CC44A	8035AHL
N8051	CC44: CC44A	8031
N8051AH	CC44: CC44A	8031
N8052AH	CC44: CC44A	8031
N8096	CC68	8096
N8096BH	CC68	8096
N8097BH	CC68: PGA\68PB	DIP64
N80C196	CC68: PGA\68PB	DIP64
N80C31BH	CC44: CC44A	8031
N80C51BH	CC44: CC44A	8031
N8344AH	CC44: CC44A	8031
N8744H	CC44: CC44A	8031
N8748H	CC44: CC44A	8035AHL
N8749H	CC44: CC44A	8035AHL
N8751H	CC44: CC44A	8031
N87C51	CC44: CC44A	8031
NE5230	DIP8	NE5230
NE5512	DIP8	AOPAMP: AOPAMPM AOPAMPPN
NE5512-SO	DIP8\SO	AOPAMP: AOPAMPM AOPAMPPN
NE5514	DIP14	AOPAMP AOPAMPPN

Part Type	PADS 2000 Decal	PADS Logic Symbol
NE5514-SO	DIP16\SOL	AOPAMP: AOPAMPM AOPAMPPN
NE5532	DIP8	AOPAMP: AOPAMPM AOPAMPPN
NE5532-SO	DIP16\SOL	AOPAMP: AOPAMPM AOPAMPPN
NE555	DIP8: DIP8\SO	NE555
NE555-SO	DIP8\SO	NE555
NE567	DIP8	NE567
P8031AH	DIP40\600	8031
P8041AH	DIP40\600	8041
P8042AH	DIP40\600	8042
P8048	DIP40\600	8035AHL
P8050	DIP40\600	8035AHL
P8050AH	DIP40\600	8035AHL
P8051AH	DIP40\600	8031
P80C31AH	DIP40\600	8031
P80C48	DIP40\600	8035AHL
P80C49	DIP40\600	8035AHL
P80C51AH	DIP40\600	8031
P80C652	DIP40\600	DIP38
P83C652	DIP40\600	DIP38
P8748	DIP40\600	8035AHL
P8749	DIP40\600	8035AHL
P87C51AH	DIP40\600	8031
P87C51FA	DIP40\600	8031
P87C652	DIP40\600	DIP38

Part Type	PADS 2000 Decal	PADS Logic Symbol
PAL10020EG8	DIP24	10H20
PAL10020EG8L	CC28: CL28. CC28A	10H20
PAL10020EV8	DIP24	10H20
PAL10020EV8L	CC28: CL28: CC28A	10H20
PAL10H20EG8	DIP24	10H20
PAL10H20EG8L	CC28: CL28: CC28A	10H20
PAL10H20EV8	DIP24	10H20
PAL10H20EV8L	CC28: CL28: CC28A	10H20
PAL16H2	DIP20: CC20: CC20A: CL20	16H2
PAL16H8	DIP20: CC20: CC20A: CL20	16H8
PAL16HD8	DIP20: CC20: CC20A: CL20	16HD8
PAL16L2	DIP20: CC20: CC20A: CL20	16L2
PAL16L8	DIP20: CC20: CC20A: CL20	16L8
PAL16LD8	DIP20: CC20. CC20A: CL20	16LD8
PAL16P2	DIP20: CC20: CC20A: CL20	16P2
PAL16P8	DIP20: CC20· CC20A: CL20	16L8
PAL16R4	DIP20: CC20. CC20A: CL20	16R4
PAL16R6	DIP20: CC20: CC20A: CL20	16R6
PAL16R8	DIP20: CC20: CC20A: CL20	16R8
PAL16RP4	DIP20: CC20: CC20A: CL20	16R4
PAL16RP6	DIP20: CC20: CC20A: CL20	16R6
PAL16RP8	DIP20: CC20: CC20A: CL20	16R8
PAL18P8	DIP20: CC20: CC20A: CL20	18P8
PAL20H8	DIP24\600	20H8

Part Type	PADS 2000 Decal	PADS Logic Symbol
PAL20L10	DIP24: DIP24\SOL	20L10
PAL20L10L	CC28: CC28A: CL28	20L10
PAL20L8	DIP24: DIP24\SOL	20L8
PAL20L8L	CC28: CC28A: CL28	20L8
PAL20R4	DIP24: DIP24\SOL	20R4
PAL20R4L	CC28: CC28A: CL28	20R4
PAL20R6	DIP24: DIP24\SOL	20R6
PAL20R6L	CC28: CC28A: CL28	20R6
PAL20R8	DIP24: DIP24\SOL	20R8
PAL20R8L	CC28: CC28A: CL28	20R8
PAL20RP10	DIP24: DIP24\SOL	20RP10
PAL20RP10L	CC28: CC28A: CL28	20RP10
PAL20RP4	DIP24: DIP24\SOL	20RP4
PAL20RP4L	CC28: CC28A: CL28	20RP4
PAL20RP6	DIP24: DIP24\SOL	20RP6
PAL20RP6L	CC28: CC28A: CL28	20RP6
PAL20RP8	DIP24: DIP24\SOL	20RP8
PAL20RP8L	CC28: CC28A: CL28	20RP8
PAL20XRP10	DIP24: DIP24\SOL	20RP10
PAL20XRP10L	CC28: CC28A: CL28	20RP10
PAL20XRP4	DIP24: DIP24\SOL	20RP4
PAL20XRP4L	CC28: CC28A: CL28	20RP4
PAL20XRP6	DIP24: DIP24\SOL	20RP6
PAL20XRP6L	CC28: CC28A: CL28	20RP6

Part Type	PADS 2000 Decal	PADS Logic Symbol
PAL20XRP8	DIP24: DIP24\SOL	20RP8
PAL20XRP8L	CC28: CC28A: CL28	20RP8
PAL22P10	DIP24: DIP24\SOL	22P10
PAL22P10L	CC28: CC28A: CL28	22P10
PAL22V10	DIP24: DIP24\SOL	22V10
PAL22V10L	CC28: CC28A: CL28	22V10
PAL22XP10	DIP24: DIP24\SOL	22XP10
PAL22XP10L	CC28: CC28A: CL28	22XP10
PAL23S8	DIP20	23S8
PALHCT29M16	DIP24	29M16
PALHCT29M16L	CC28: CC28A: CL28	29M16
PGA\132P	PGA\132P	GEN\DIP132
PGA\68P	PGA\68P: CC68A: PGA\68PC	GEN\DIP68
PGA\68PA	PGA\68PA	GEN\DIP68
PGA\68PB	PGA\68PB: CC68	GEN\DIP68
PGA\68PC	PGA\68PC: CC68	GEN\DIP68
PGA\84P	PGA\84P	GEN\DIP84
PGA\88P	PGA\88P	GEN\DIP88
PH2955T	TO-220	NPN-V: NPN-VA: NPN-H: NPN-HA
PH3055T	TO-220	NPN-V: NPN-VA: NPN-H: NPN-HA
PLC153	DIP20: CC20: CC20A	PLC153
PLC20V8	DIP24	20V8
PLC473	DIP24	PLC473
PLHS153	DIP20: CC20: CC20A	PLC153

Part Type	PADS 2000 Decal	PADS Logic Symbol
PLHS18P8A	DIP20: CC20: CC20A	18P8
PLHS473	DIP24	PLC473
PLHS473A	DIP24	PLC473
PLHS501A	CC52	PLHS501
PLHS502A	CC52	PLHS502
PLMC153	DIP20: CC20: CC20A	PLC153
PLS100	DIP28\600: CC28: CC28A	PLS100
PLS101	DIP28\600: CC28: CC28A	PLS100
PLS103	DIP28\600: CC28: CC28A	PLS103
PLS105	DIP28\600: CC28: CC28A	PLS105
PLS105A	DIP28\600: CC28: CC28A	PLS105
PLS151	DIP20: CC20: CC20A	PLS151
PLS153	DIP20: CC20: CC20A	PLC153
PLS153A	DIP20: CC20: CC20A	PLS153
PLS155	DIP20: CC20: CC20A	PLS155
PLS157	DIP20: CC20: CC20A	PLS157
PLS159	DIP20: CC20: CC20A	PLS159
PLS159A	DIP20: CC20: CC20A	PLS159
PLS161	DIP24	PLS161
PLS162	DIP24	PLS162
PLS163	DIP24	PLS163
PLS167	DIP24	PLS167
PLS167A	DIP24	PLS167
PLS168	DIP24	PLS168

Part Type	PADS 2000 Decal	PADS Logic Symbol
PLS168A	DIP24	PLS168
PLS173	DIP24	PLS173
PLS173-CC	CC28: CC28A	PLS173
PLS173A	CC28: CC28A: CL20	PLS173
PLS179	DIP24	PLS179
PLS179A	DIP24	PLS179
PLUS153	DIP20: CC20: CC20A	PLC153
PLUS173	DIP24	PLS173
PLUS173A	DIP24	PLS173
PLUS405A	DIP28\600	PLUS405
PLUS405A-CC	CC28: CC28A	PLUS405
PN2222	TO-92	NPN-V: NPN-VA: NPN-H: NPN-HA
PN2907	TO-92	PNP-V: PNP-VA: PNP-H: PNP-HA
R1/2W	R1/2W	RESZ-H: RESZ-V: RESB-H: RESB-V
R1/4W	R1/4W: R1/4WA: R1/4WB	RESZ-H: RESZ-V: RESB-H: RESB-V
R1/8W	R1/8W: R1/8WA: R1/8WB	RESZ-H: RESZ-V: RESB-H: RESB-V
R1W	R1W	RESZ-H: RESZ-V: RESB-H: RESB-V
R2W	R2W	RESZ-H: RESZ-V. RESB-H: RESB-V
R3W	R3W	RESZ-H: RESZ-V: RESB-H: RESB-V
RDIP147RES	DIP14	RESZ-H: RESZ-V: RESB-H: RESB-V
RDIP168RES	DIP16	RESZ-H: RESZ-V: RESB-H: RESB-V
RES	R1/8W: R1/8WA	RESZ-H: RESZ-V: RESB-H: RESB-V
RESSMD-1206	CC1206	RESZ-H: RESZ-V: RESB-H: RESB-V
RLY-DPDT	PCB-NULL	RLY-DPDT

Part Type	PADS 2000 Decal	PADS Logic Symbol
RLY-MZ-SPDT	RLY-MZ	RLY-SPDT
RLY-RLY11-DPDT	PCB-NULL	RLY-DPDT
RLY-RLY11-SPDT	PCB-NULL	RLY-SPDT
RLY-RY-DPDT	RLY-RY	RLY-SW-SPDT: RLY-COIL
RLY-SPDT	RLY-MZ	RLY-SPDT
RSIP10P5R	SIP\10P	RESZ-H: RESZ-V: RESB-H: RESB-V
RSIP10P9R	SIP\10P	RESZ-H: RESZ-V: RESB-H: RESB-V: RESZ-H1P: RESZ-V1P: RESB-H1P: RESB-V1P
RSIP10P9R+5	SIP\10P	RESZ-H5: RESZ-V5: RESB-H5: RESB-V5
RSIP10P9RG	SIP\10P	RESZ-HG: RESZ-VG: RESB-HG: RESB-VG
RSIP8P4R	SIP\8P	RESZ-H: RESZ-V: RESB-H: RESB-V
RSIP8P7R	SIP\8P	RESZ-H: RESZ-V: RESB-H: RESB-V: RESZ-H1P: RESZ-V1P: RESB-H1P: RESB-V1P
RSIP8P7R+5	SIP\8P	RESZ-H5: RESZ-V5: RESB-H5: RESB-V5
RSIP8P7RG	SIP\8P	RESZ-HG: RESZ-VG: RESB-HG: RESB-VG
SC60	CAP\CR08	CAP: CAPA: CAPH: CAPAH
SIP\10P	SIP\10P	
SIP\11P	SIP\11P	
SIP\12P	SIP\12P	
SIP\13P	SIP\13P	
SIP\14P	SIP\14P	
SIP\15P	SIP\15P	
SIP\16P	SIP\16P	
SIP\17P	SIP\17P	
SIP\18P	SIP\18P	
SIP\19P	SIP\19P	

Part Type	PADS 2000 Decal	PADS Logic Symbol
SIP\1P	SIP\1P	
SIP\20P	SIP\20P	
SIP\2P	SIP\2P	
SIP\3P	SIP\3P	
SIP\4P	SIP\4P	
SIP\5P	SIP\5P	
SIP\6P	SIP\6P	
SIP\7P	SIP\7P	
SIP\8P	SIP\8P	
SIP\9P	SIP\9P	
SW-SPDT	PCB-NULL	SW-SPDT
SW-SPST-NC	PCB-NULL	SW-SPST-NC
SW-SPST-NO	PCB-NULL	SW-SPST-NO
TCA5600	DIP18	TCA5600
TCF5600	DIP18	TCA5600
THCT1010	DIP64\900	THCT1010
THCT1010\CC	CC68	THCT1010
THCT2000E	DIP28\600	CC28: CC28A THCT2000E
THCT2000M	DIP28\600	CC28: CC28A THCT2000M
THCT29520	DIP24: DIP24\SOL	29520
THCT29521	DIP24: DIP24\SOL	29521
TIP2955	SOT93	NPN-V: NPN-VA: NPN-H: NPN-HA
TIP3055	SOT93	NPN-V: NPN-VA: NPN-H: NPN-HA
TO-100	TO-100	

Part Type	PADS 2000 Decal	PADS Logic Symbol
TO-18	TO-18	
TO-220	TO-220	
TO-220\UP	TO-220\UP	
TO-220\UP\4P	TO-220\UP\4P	
TO-39	TO-39	
TO-5	TO-5	
TO-71	TO-71	
TO-78	TO-78	
TO-8	TO-8	
TO-92	TO-92	
TO-99	TO-99	
TRIAC-TO220	TO-220	TRIAC-V
TRIAC-TO92-KAG	TO-92	TRIAC-V
TRJFETN-TO92	TO-92	JFETN-V
TRJFETP-TO92	TO-92	JFETP-V
TRMFETN-TO220	TO-220	MFETN-V
TRMFETN-TO92	TO-92	MFETN-V
TRMFETP-TO220	TO-220	MFETP-V
TRMFETP-TO92	TO-92	MFETP-V
TRNPN-SOT93	SOT93	NPN-V: NPN-VA: NPN-H: NPN-HA
TRNPN-TO18-EBC	TO-18	NPN-V: NPN-VA: NPN-H: NPN-HA
TRNPN-TO220	TO-220	NPN-V: NPN-VA: NPN-H: NPN-HA
TRNPN-TO3	TO-3	NPN-V: NPN-VA: NPN-H: NPN-HA
TRNPN-TO39	TO-39	NPN-V: NPN-VA: NPN-H: NPN-HA

Part Type	PADS 2000 Decal	PADS Logic Symbol
TRNPN-TO92-BCE	TO-92	NPN-V: NPN-VA: NPN-H: NPN-HA
TRNPN-TO92-BEC	TO-92	NPN-V: NPN-VA: NPN-H: NPN-HA
TRNPN-TO92-CBE	TO-92	NPN-V: NPN-VA: NPN-H: NPN-HA
TRNPN-TO92-EBC	TO-92	NPN-V: NPN-VA: NPN-H: NPN-HA
TRPNP-SOT93	SOT93	PNP-V: PNP-VA: PNP-H: PNP-HA
TRPNP-TO18-EBC	TO-18	PNP-V: PNP-VA: PNP-H: PNP-HA
TRPNP-TO220	TO-220	PNP-V: PNP-VA: PNP-H: PNP-HA
TRPNP-TO3	TO-3	PNP-V: PNP-VA: PNP-H: PNP-HA
TRPNP-TO39	TO-39	PNP-V: PNP-VA: PNP-H: PNP-HA
TRPNP-TO92-BCE	TO-92	PNP-V: PNP-VA: PNP-H: PNP-HA
TRPNP-TO92-BEC	TO-92	PNP-V: PNP-VA: PNP-H: PNP-HA
TRPNP-TO92-CBE	TO-92	PNP-V: PNP-VA: PNP-H: PNP-HA
TRPNP-TO92-EBC	TO-92	PNP-V: PNP-VA: PNP-H: PNP-HA
TRSCR-TO220	TO-220	SCR-V
TRSCR-TO92-AGK	TO-92	SCR-V
TRSCR-TO92-KGA	TO-92	SCR-V
VRES	VRES	VRESZ-H: VRESZ-V
VRES\TOP\ADJ	VRES\TOP\ADJ	VRESZ-H: VRESZ-V
XTAL1	XTAL1	XTALH
XTAL2	XTAL2	XTALH
\00	DIP14: DIP14\SO	NAND2: NANDDM
\00-CC	CC20: CC20A	NAND2: NANDDM
\00A	DIP14: DIP14\SO	NAND2: NANDDM
\01	DIP14: DIP14\SO	NAND2OC: NANDDMOC

Part Type	PADS 2000 Decal	PADS Logic Symbol
\02	DIP14: DIP14\SO	NOR2: NORDM
\02-CC	CC20: CC20A	NOR2: NORDM
\03	DIP14: DIP14\SO	NAND2OC: NANDDMOC
\03-CC	CC20: CC20A	NAND2OC: NANDDMOC
\03A	DIP14: DIP14\SO	NAND2OC: NANDDMOC
\04	DIP14: DIP14\SO	INV: INVL: INVDM: INVDML
\04-CC	CC20: CC20A	INV: INVL: INVDM: INVDML
\04A	DIP14: DIP14\SO	INV: INVL: INVDM: INVDML
\05	DIP14: DIP14\SO	INVOC: INVDMOC
\05-CC	CC20: CC20A	INVOC: INVDMOC
\05A	DIP14: DIP14\SO	INVOC: INVDMOC
\06	DIP14: DIP14\SO	INVOC: INVDMOC
\07	DIP14: DIP14\SO	INVNONOC
\08	DIP14: DIP14\SO	AND2
\08-CC	CC20: CC20A	AND2
\09	DIP14: DIP14\SO	AND2OC
\09-CC	CC20: CC20A	AND2OC
\10	DIP14: DIP14\SO	NAND3
\10-CC	CC20: CC20A	NAND3
\1000	DIP14: DIP14\SO	NAND2: NANDDM
\1000-CC	CC20: CC20A	NAND2: NANDDM
\1000A	DIP14: DIP14\SO	NAND2: NANDDM
\1002-CC	CC20: CC20A	NOR2: NORDM
\1002A	DIP14: DIP14\SO	NOR2: NORDM

Part Type	PADS 2000 Decal	PADS Logic Symbol
\1003-CC	CC20:CC20A	NAND2:NANDDM
\1003A	DIP14:DIP14\SO	NAND2:NANDDM
\1004	DIP14:DIP14\SO	INV:INVL:INVDM:INVDML
\1004-CC	CC20:CC20A	INV:INVL:INVDM:INVDML
\1005	DIP14:DIP14\SO	INV:INVL:INVDM:INVDML
\1005-CC	CC20:CC20A	INV:INVL:INVDM:INVDML
\1008	DIP14:DIP14\SO	AND2
\1008-CC	CC20:CC20A	AND2
\1008A	DIP14:DIP14\SO	AND2
\1010A	DIP14:DIP14\SO	NAND3
\1010A-CC	CC20:CC20A	NAND3
\1011A	DIP14:DIP14\SO	AND3
\1011A-CC	CC20:CC20A	AND3
\1020A	DIP14:DIP14\SO	NAND4
\1020A-CC	CC20:CC20A	NAND4
\1032	DIP14:DIP14\SO	OR2
\1032A	DIP14:DIP14\SO	OR2
\1032A-CC	CC20:CC20A	OR2
\1034	DIP14	INV:INVL:INVDM:INVDML
\1035	DIP14	INVNONOC
\1036	DIP14:DIP14\SO	NOR2:NORDM
\107	DIP14:DIP14\SO	\107
\107-CC	CC20:CC20A	\107
\109	DIP16:DIP16\SO	\109

Part Type	PADS 2000 Decal	PADS Logic Symbol
\109-CC	CC20: CC20A	\109
\11	DIP14: DIP14\SO	AND3
\11-CC	CC20: CC20A	AND3
\11000	DIP16: DIP16\SO	NAND2: NANDDM
\11002	DIP16: DIP16\SO	NOR2: NORDM
\11004	DIP20: DIP20\SOL	INV: INVL: INVDM: INVDML
\11008	DIP16: DIP16\SO	AND2
\11010	DIP16: DIP16\SO	NAND3
\11011	DIP16: DIP16\SO	AND3
\11013	DIP14: DIP14\SO	NAND4
\11014	DIP16: DIP16\SO	INVST
\11020	DIP14: DIP14\SO	NAND4
\11021	DIP14: DIP14\SO	AND4
\11027	DIP16: DIP16\SO	NOR3
\11030	DIP14: DIP14\SO	NAND8
\11032	DIP16: DIP16\SO	OR2
\11034	DIP20: DIP20\SOL	INVNON: INVNONL
\11074	DIP14: DIP14\SO	\74
\11086	DIP16: DIP16\SO	XOR
\11109	DIP16: DIP16\SO	\109
\11112	DIP16: DIP16\SO	\112
\11132	DIP16: DIP16\SO	NANDST
\11138	DIP16: DIP16\SO	\138
\11151	DIP16: DIP16\SO	\151

Part Type	PADS 2000 Decal	PADS Logic Symbol
\11158	DIP20:DIP20\SOL	\158
\11160	DIP20:DIP20\SOL	\160
\11162	DIP20:DIP20\SOL	\160
\11175	DIP20:DIP20\SOL	\175
\11190	DIP20:DIP20\SOL	\190
\11191	DIP20:DIP20\SOL	\190
\11194	DIP20:DIP20\SOL	\194
\112	DIP16:DIP16\SO	\112
\112-CC	CC20	\112
\11238	DIP16:DIP16\SO	\238
\11240	DIP24:DIP24\SOL	\240
\11241	DIP24:DIP24\SOL	\244
\11244	DIP24:DIP24\SOL	\244
\11245	DIP24:DIP24\SOL	\573
\11251	DIP16:DIP16\SO	\151
\11253	DIP16:DIP16\SO	\153
\11257	DIP20:DIP20\SOL	\157
\11258	DIP20:DIP20\SOL	\158
\11280	DIP14:DIP14\SO	\280
\112A	DIP16:DIP16\SO	\112
\113	DIP14:DIP14\SO	\113
\113-CC	CC20	\113
\11353	DIP16:DIP16\SO	\352
\11373	DIP24:DIP24\SOL	\573

Part Type	PADS 2000 Decal	PADS Logic Symbol
\11374	DIP24:DIP24\SOL	\574
\113A	DIP14:DIP14\SO	\113
\114	DIP14:DIP14\SO	\114
\114-CC	CC20	\114
\11520	DIP20:DIP20\SOL	\521
\11521	DIP20:DIP20\SOL	\521
\11533	DIP24:DIP24\SOL	\563
\11534	DIP24:DIP24\SOL	\564
\11640	DIP24:DIP24\SOL	\573
\11810	DIP16:DIP16\SO	XNOR
\12	DIP14:DIP14\SO	NAND3
\123	DIP16:DIP16\SO	\221
\1240	DIP20	\240
\1241	DIP20	\244 \244A
\1242	DIP14:DIP14\SO	\1242
\1243	DIP14:DIP14/SO	\1243
\1244	DIP20	\244
\1244A	DIP20:CC20	\244
\1245	DIP20	\573
\125	DIP14:DIP14\SO	\125:\125A
\125-CC	CC20:CC20A	\125:\125A
\126	DIP14:DIP14\SO	\126:\126A
\126-CC	CC20:CC20A	\126:\126A
\128	DIP14:DIP14\SO	NOR2

Part Type	PADS 2000 Decal	PADS Logic Symbol
\13	DIP14:DIP14\SO	NAND4
\131	DIP16	\131
\132	DIP14:DIP14\SO	NANDST
\133	DIP16:DIP16\SO	NAND13
\133-CC	CC20:CC20A	NAND13
\134	DIP16:DIP16\SO	\134
\135	DIP16:DIP16\SO	\135
\136	DIP14:DIP14\SO	XOROC
\137	DIP16:DIP16\SO	\137
\137-CC	CC20:CC20A	\137
\138	DIP16:DIP16\SO	\138:\138S
\138-CC	CC20:CC20A	\138
\139	DIP16:DIP16\SO	\139:\139S
\139-CC	CC20:CC20A	\139
\14	DIP14:DIP14\SO	INVST
\14-CC	CC20:CC20A	INVST
\140	DIP14:DIP14\SO	NAND4
\145	DIP16	\145
\147	DIP16:DIP16\SO	\147
\148	DIP16:DIP16\SO	\148
\15	DIP14:DIP14\SO	AND3
\150	DIP24\600	\150
\151	DIP16:DIP16\SO	\151
\151-CC	CC20:CC20A	\151

Part Type	PADS 2000 Decal	PADS Logic Symbol
\151A	DIP16: DIP16\SO	\151
\152	DIP14	\152
\152A	DIP14	\152
\153	DIP16: DIP16\SO	\153
\153-CC	CC20: CC20A	\153
\154	DIP24\600: DIP24\SOL	\154
\154-CC	CC28: CC28A	\154
\155	DIP16	\155
\156	DIP16	\156
\157	DIP16: DIP16\SO	\157
\157-CC	CC20: CC20A	\157
\158	DIP16: DIP16\SO	\158
\158-CC	CC20: CC20A	\158
\16	DIP14: DIP14\SO	INVOC: INVDMOC
\160	DIP16: DIP16\SO	\160
\160-CC	CC20: CC20A	\160
\160A	DIP16: DIP16\SO	\160
\161	DIP16: DIP16\SO	\160
\161-CC	CC20: CC20A	\160
\161A	DIP16: DIP16\SO	\160
\162	DIP16: DIP16\SO	\160
\162-CC	CC20: CC20A	\160
\1620	DIP20	\1620
\1621	DIP20	\1620

Part Type	PADS 2000 Decal	PADS Logic Symbol
\1622	DIP20	\1620
\1623	DIP20	\1620
\162A	DIP16:DIP16\SO	\160
\163	DIP16:DIP16\SO	\160
\163-CC	CC20:CC20A	\160
\1638	DIP20	\1620
\1639	DIP20	\1620
\163A	DIP16:DIP16\SO	\160
\164	DIP14:DIP14\SO	\164
\164-CC	CC20:CC20A	\164
\1640A	DIP20	\1620
\1641	DIP20	\1620
\1642	DIP20	\1620
\1643	DIP20	\1620
\1644	DIP20	\1620
\1645	DIP20	\1620
\165	DIP16:DIP16\SO	\165
\165-CC	CC20:CC20A	\165
\166	DIP16:DIP16\SO	\166
\166-CC	CC20:CC20A	\166
\168	DIP16:DIP16\SO	\168
\168A	DIP16:DIP16\SO	\168
\169	DIP16:DIP16\SO	\168
\17	DIP14:DIP14\SO	INVNONOC

Part Type	PADS 2000 Decal	PADS Logic Symbol
\170	DIP16:DIP16\SO	\170
\173	DIP16:DIP16\SO	\173
\174	DIP16:DIP16\SO	\174
\174-CC	CC20:CC20A	\174
\175	DIP16:DIP16\SO	\175
\175-CC	CC20:CC20A	\175
\180	DIP14:DIP14\SO	\180
\180-CC	CC20:CC20A	\180
\181	DIP24:DIP24\SOL	\181
\181-CC	CC28:CC28A	\181
\182	DIP16	\182
\189	DIP16	\189
\190	DIP16:DIP16\SO	\190
\190-CC	CC20:CC20A	\190
\191	DIP16:DIP16\SO	\190
\191-CC	CC20:CC20A	\190
\192	DIP16:DIP16\SO	\192
\192-CC	CC20:CC20A	\192
\193	DIP16:DIP16\SO	\192
\193-CC	CC20:CC20A	\192
\194	DIP16:DIP16\SO	\194
\194-CC	CC20:CC20A	\194
\194A	DIP16:DIP16\SO	\194
\195	DIP16:DIP16\SO	\195

Part Type	PADS 2000 Decal	PADS Logic Symbol
\195-CC	CC20: CC20A	\195
\195A	DIP16: DIP16\SO	\195
\198	DIP24: DIP24\SOL	\198
\199	DIP24: DIP24\SOL	\199
\20	DIP14: DIP14\SO	NAND4
\20-CC	CC20: CC20A	NAND4
\21	DIP14: DIP14\SO	AND4
\21-CC	CC20: CC20A	AND4
\219A	DIP16	\219
\22	DIP14	NAND4
\221	DIP16: DIP16\SO	\221
\222	DIP20	\222
\224	DIP16	\224
\225	DIP20	\225
\22A	DIP14: DIP14\SO	NAND4
\230	DIP20	\230
\231	DIP20	\231
\232A	DIP16: DIP16\SO: DIP16\SOL	\232A
\232A-CC	CC20: CC20A	\232A
\233A	DIP20: DIP20\SOL	\233A
\233A-CC	CC20: CC20A	\233A
\237	DIP16: DIP16\SO	\237
\237-CC	CC20: CC20A	\237
\238	DIP16: DIP16\SO	\238

Part Type	PADS 2000 Decal	PADS Logic Symbol
\238-CC	CC20: CC20A	\238
\239	DIP16: DIP16\SO	\239: \239S
\239-CC	CC20: CC20A	\239
\240	DIP20: DIP20\SOL	\240
\240-CC	CC20: CC20A	\240
\240A	DIP20: DIP20\SOL	\240
\241	DIP20: DIP20\SOL	\244: \241
\241-CC	CC20: CC20A	\244: \241
\241A	DIP20: DIP20\SOL	\244: \241
\242	DIP14: DIP14\SO	\242
\242-CC	CC20: CC20A	\242
\242A	DIP14: DIP14\SO	\242
\243	DIP14: DIP14\SO	\242
\243-CC	CC20: CC20A	\242
\243A	DIP14: DIP14\SO	\242
\244	DIP20: DIP20\SOL	\244
\244-CC	CC20: CC20A	\244
\244A	DIP20: DIP20\SOL	\244
\245	DIP20: DIP20\SOL	\573
\245-CC	CC20: CC20A	\573
\245A	DIP20: DIP20\SOL	\573
\247	DIP16	\247
\248	DIP16	\248
\249	DIP16	\248

Part Type	PADS 2000 Decal	PADS Logic Symbol
\250	DIP24\600	\250
\250-CC	CC28: CC28A	\250
\251	DIP16: DIP16\SO	\151
\251-CC	CC20: CC20A	\151
\253	DIP16: DIP16\SO	\153
\253-CC	CC20: CC20A	\153
\257	DIP16: DIP16\SO	\157
\257-CC	CC20: CC20A	\157
\258	DIP16: DIP16\SO	\158
\258-CC	CC20: CC20A	\158
\259	DIP16: DIP16\SO	\259
\259-CC	CC20: CC20A	\259
\26	DIP14: DIP14\SO	NAND2OC: NANDOMOC
\260	DIP14: DIP14\SO	NOR5
\2620	DIP20: DIP20\SOL	\573
\2623	DIP20: DIP20\SOL	\573
\264	DIP16: DIP16\SO	\264
\2640	DIP20: DIP20\SOL	\573
\2645	DIP20: DIP20\SOL	\573
\266	DIP14: DIP14\SO	XNOR
\266-CC	CC20: CC20A	XNOR
\27	DIP14: DIP14\SO	NOR3
\27-CC	CC20: CC20A	NOR3
\273	DIP20: DIP20\SOL	\574

Part Type	PADS 2000 Decal	PADS Logic Symbol
\273-CC	CC20: CC20A	\574
\28	DIP14: DIP14\SO	NOR2: NORDM
\280	DIP14: DIP14\SO	\280
\280-CC	CC20: CC20A	\280
\282	DIP20	\282
\283	DIP16: DIP16\SO	\283
\283-CC	CC20: CC20A	\283
\286	DIP14: DIP14\SO	\286
\286-CC	CC20: CC20A	\286
\28A	DIP14: DIP14\SO	NOR2: NORDM
\290	DIP14: DIP14\SO	\290
\293	DIP14: DIP14\SO	\293
\298	DIP16: DIP16\SO	\298
\298-CC	CC20: CC20A	\298
\299	DIP20: DIP20\SOL	\299
\299-CC	CC20: CC20A	\299
\30	DIP14: DIP14\SO	NAND8
\30-CC	CC20: CC20A	NAND8
\32	DIP14: DIP14\SO	OR2
\32-CC	CC20: CC20A	OR2
\322	DIP20: DIP20\SOL	\322
\322-CC	CC20: CC20A	\322
\323	DIP20: DIP20\SOL	\299
\323-CC	DIP20: DIP20\SOL	\299

Part Type	PADS 2000 Decal	PADS Logic Symbol
\324	DIP14: DIP14\SO	AOPAMPM
\33	DIP14: DIP14\SO	NOR2OC: NORDMOC
\33A	DIP14: DIP14\SO	NOR2OC: NORDMOC
\34	DIP14: DIP14\SO	INVNON: INVNONL
\35	DIP14: DIP14\SO	INVNON: INVNONL
\352	DIP16: DIP16\SO	\352
\352-CC	CC20: CC20A	\352
\353	DIP16: DIP16\SO	\352
\353-CC	CC20: CC20A	\352
\354	DIP20: DIP20\SOL	\354
\354-CC	CC20: CC20A	\354
\356	DIP20: DIP20\SOL	\356
\356-CC	CC20: CC20A	\356
\363	DIP20: DIP20\SOL	\573
\364	DIP20: DIP20\SOL	\574
\365	DIP16: DIP16\SO	\365
\365-CC	CC20: CC20A	\365
\366	DIP16: DIP16\SO	\366
\366-CC	CC20: CC20A	\366
\367	DIP16: DIP16\SO	\367A: \367B
\367-CC	CC20: CC20A	\367A: \367B
\368	DIP16: DIP16\SO	\368A: \368B
\368-CC	CC20: CC20A	\368A: \368B
\37	DIP14: DIP14\SO	NAND2: NANDM

Part Type	PADS 2000 Decal	PADS Logic Symbol
\373	DIP20: DIP20\SOL	\573
\373-CC	CC20: CC20A	\573
\374	DIP20: DIP20\SOL	\574
\374-CC	CC20: CC20A	\574
\375	DIP16: DIP16\SO	\75
\377	DIP20: DIP20\SOL	\574
\377-CC	CC20: CC20A	\574
\378	DIP16: DIP16\SO	\174
\379	DIP16: DIP16\SO	\175
\37A	DIP14: DIP14\SO	NAND2: NANDDM
\38	DIP14: DIP14\SO	NAND2OC: NANDDMOC
\381	DIP20	\381
\386	DIP14: DIP14\SO	XOR
\386-CC	CC20: CC20A	XOR
\38A	DIP14: DIP14\SO	NAND2OC: NANDDMOC
\39	DIP14: DIP14\SO	NAND2OC: NANDDMOC
\390	DIP16: DIP16\SO	\390
\390-CC	CC20: CC20A	\390
\393	DIP14: DIP14\SO	\393
\393-CC	CC20: CC20A	\390
\395	DIP16: DIP16\SO	\395
\395-CC	CC20: CC20A	\395
\395A	DIP16: DIP16\SO	\395
\40	DIP14: DIP14\SO	NAND4

Part Type	PADS 2000 Decal	PADS Logic Symbol
\4002	DIP14:DIP14\SO	NOR4
\4016	DIP14:DIP14\SO	4066:4066V
\4040	DIP16:DIP16\SO	\4040
\4046	DIP16:DIP16\SO	\4046
\4049	DIP16:DIP16\SO	INV:INVL:INVDM:INVDML
\4051	DIP16:DIP16\SO	4051
\4052	DIP16:DIP16\SO	4052
\4053	DIP16:DIP16\SO	\4053
\4060	DIP16:DIP16\SO	\4060
\4061	DIP16:DIP16\SO	4060
\4066	DIP14:DIP14\SO	4066:4066V
\4075	DIP14:DIP14\SO	OR3
\4094	DIP16:DIP16\SO	4094
\40A	DIP14:DIP14\SO	NAND4
\412	DIP24:DIP24\SOL:DIP24\300	\412
\42	DIP16:DIP16\SO	\42
\42A	DIP16:DIP16\SO	\42
\4316	DIP16:DIP16\SO	\4316
\432	DIP24:DIP24\SOL	\432
\4351	DIP20:DIP20\SOL	4351
\4352	DIP20:DIP20\SOL	4352
\4353	DIP20:DIP20\SOL	4353
\4511	DIP16:DIP16\SO	4511
\4518	DIP16:DIP16\SO	\4518

Part Type	PADS 2000 Decal	PADS Logic Symbol
\4520	DIP16:DIP16\SO	\4518
\4538	DIP16:DIP16\SO	\4538
\465A	DIP20:DIP20\SOL	\541
\466A	DIP20:DIP20\SOL	\540
\467A	DIP20:DIP20\SOL	\244
\468A	DIP20:DIP20\SOL	\240
\47	DIP16	\47
\470	DIP20	\470
\471	DIP20	\470
\472	DIP20	\472
\4724	DIP16:DIP16\SO	4724
\474	DIP24\600	\474
\475	DIP24\600	\475
\47A	DIP16	\47
\49	DIP14	\49
\490	DIP16:DIP16\SO	\490
\490-CC	CC20:CC20A	\390
\51	DIP14:DIP14\SO	\51A:\51B
\51-CC	CC20:CC20A	\51A:\51B
\518	DIP20	\518
\519	DIP20	\518
\520	DIP20	\520
\521	DIP20	\520
\522	DIP20	\520

Part Type	PADS 2000 Decal	PADS Logic Symbol
\526	DIP20	\526
\527	DIP20	\527
\528	DIP16	\528
\533	DIP20: DIP20\SOL	\563
\533-CC	CC20: CC20A	\563
\534	DIP20: DIP20\SOL	\564
\534-CC	CC20: CC20A	\564
\538	DIP20: DIP20\SOL	\538
\539	DIP20: DIP20\SOL	\539
\540	DIP20: DIP20\SOL	\540
\540-CC	CC20: CC20A	\540
\541	DIP20: DIP20\SOL	\541
\541-CC	CC20: CC20A	\541
\560A	DIP20	\560A
\561A	DIP20	\560A
\563	DIP20: DIP20\SOL	\563
\563-CC	CC20: CC20A	\563
\564	DIP20: DIP20\SOL	\564
\564-CC	CC20: CC20A	\564
\568	DIP20: DIP20\SOL	\568
\568A	DIP20: DIP20\SOL	\568
\569	DIP20: DIP20\SOL	\568
\569A	DIP20: DIP20\SOL	\568
\573	DIP20: DIP20\SOL	\573

Part Type	PADS 2000 Decal	PADS Logic Symbol
\573-CC	CC20: CC20A	\573
\574	DIP20: DIP20\SOL	\574
\574-CC	CC20: CC20A	\574
\575	DIP24	\575
\576	DIP20	\576
\577	DIP24	\577
\580	DIP20	\580
\589	DIP16	\589
\590	DIP16: DIP16\SO	\590
\590-CC	CC20: CC20A	\590
\594	DIP16: DIP16\SO	\594
\594-CC	CC20: CC20A	\590
\595	DIP16: DIP16\SO	\594
\595-CC	CC20: CC20A	\590
\604	DIP28\600	\604
\604-CC	CC28: CC28A	\604
\610	DIP40\600	\610
\610-CC	CC44: CC44A	\610
\611	DIP40\600	\610
\611-CC	CC44: CC44A	\610
\612	DIP40\600	\610
\612-CC	CC44: CC44A	\610
\613	DIP40\600	\610
\613-CC	CC44: CC44A	\610

Part Type	PADS 2000 Decal	PADS Logic Symbol
\616	DIP40\600	\616
\616-CC	CC28: CC28A	\616
\617	DIP40\600	\616
\617-CC	CC28: CC28A	\616
\620	DIP20: DIP20\SOL	\573
\620-CC	CC20: CC20A	\573
\620A	DIP20: DIP20\SOL	\573
\621	DIP20: DIP20\SOL	\573
\621A	DIP20: DIP20\SOL	\573
\622	DIP20: DIP20\SOL	\573
\622A	DIP20: DIP20\SOL	\573
\623	DIP20: DIP20\SOL	\573
\623-CC	CC20: CC20A	\573
\623A	DIP20: DIP20\SOL	\573
\6301	DIP52\600	\6301
\6302	DIP52\600	\6302
\632	DIP52\600	\632
\632-CC	CC68: CC68A	\632
\632A	DIP52\600	\632
\632A-CC	CC68: CC68A	\632
\633	DIP52\600	\632
\633-CC	CC68: CC68A	\632
\634	DIP48\600	\634
\634-CC	CC68: CC68A	\634

Part Type	PADS 2000 Decal	PADS Logic Symbol
\635	DIP48\600	\634
\635-CC	CC68: CC68A	\634
\638	DIP20: DIP20\SOL	\573
\638A	DIP20: DIP20\SOL	\573
\639	DIP20: DIP20\SOL	\573
\639A	DIP20: DIP20\SOL	\573
\64	DIP14	\64
\640	DIP20: DIP20\SOL	\573
\640-CC	CC20: CC20A	\573
\640A	DIP20: DIP20\SOL	\573
\641	DIP20: DIP20\SOL	\573
\641A	DIP20: DIP20\SOL	\573
\642	DIP20: DIP20\SOL	\573
\642A	DIP20: DIP20\SOL	\573
\643	DIP20: DIP20\SOL	\573
\643-CC	CC20: CC20A	\573
\643A	DIP20: DIP20\SOL	\573
\644	DIP20: DIP20\SOL	\573
\644A	DIP20: DIP20\SOL	\573
\645	DIP20: DIP20\SOL	\573
\645-CC	CC20: CC20A	\573
\645A	DIP20: DIP20\SOL	\573
\646	DIP24: DIP24\SOL	\646
\646-CC	CC28: CC28A	\646

Part Type	PADS 2000 Decal	PADS Logic Symbol
\647	DIP24:DIP24\SOL	\646
\647-CC	CC28:CC28A	\646
\648	DIP24:DIP24\SOL	\646
\648-CC	CC28:CC28A	\646
\649	DIP24:DIP24\SOL	\646
\649-CC	CC28:CC28A	\646
\651	DIP24:DIP24\SOL:DIP24\300	\646
\651-CC	CC28:CC28A	\646
\652	DIP24:DIP24\SOL:DIP24\300	\646
\652-CC	CC28:CC28A	\646
\653	DIP24:DIP24\SOL:DIP24\300	\646
\653-CC	CC28:CC28A	\646
\654	DIP24:DIP24\SOL.DIP24\300	\646
\654-CC	CC28:CC28A	\646
\658	DIP24:DIP24\SOL	\658
\658-CC	CC28:CC28A	\658
\659	DIP24:DIP24\SOL	\658
\659-CC	CC28:CC28A	\658
\664	DIP24:DIP24\SOL	\658
\664-CC	CC28:CC28A	\658
\665	DIP24:DIP24\SOL	\658
\665-CC	CC28:CC28A	\658
\666	DIP24:DIP24\300:DIP24\SOL	\666
\666-CC	CC28:CC28A	\666

Part Type	PADS 2000 Decal	PADS Logic Symbol
\667	DIP24:DIP24\SOL:DIP24\300	\667
\667-CC	CC28:CC28A	\667
\670	DIP16 ,	\670
\673	DIP24:DIP24\SOL	\673
\674	DIP24:DIP24\SOL	\674
\675	DIP24:DIP24\SOL	\675
\676	DIP24:DIP24\SOL	\674
\677	DIP24:DIP24\SOL	\677
\677-CC	CC28:CC28A	\677
\678	DIP24:DIP24\SOL	\678
\678-CC	CC28:CC28A	\678
\679	DIP20:DIP20\SOL	\679
\679-CC	CC20:CC20A	\679
\680	DIP20:DIP20\SOL	\680
\680-CC	CC20:CC20A	\679
\682	DIP20:DIP20\SOL	\682
\682-CC	CC20:CC20A	\682
\684	DIP20:DIP20\SOL	\682
\684-CC	CC20:CC20A	\682
\686	DIP24\300:DIP24\SOL	\686
\686-CC	CC28:CC28A	\686
\688	DIP20:DIP20\SOL	\521
\688-CC	CC20:CC20A	\521
\689	DIP20:DIP20\SOL	\521

Part Type	PADS 2000 Decal	PADS Logic Symbol
\690	DIP20:DIP20\SOL	\690
\690-CC	CC20:CC20A	\690
\691	DIP20:DIP20\SOL	\690
\691-CC	CC20:CC20A	\690
\692	DIP20:DIP20\SOL	\690
\693	DIP20:DIP20\SOL	\690
\693-CC	CC20:CC20A	\690
\696	DIP20:DIP20\SOL	\696
\696-CC	CC20:CC20A	\696
\697	DIP20:DIP20\SOL	\696
\697-CC	CC20:CC20A	\696
\698	DIP20:DIP20\SOL	\696
\698-CC	CC20:CC20A	\696
\699	DIP20:DIP20\SOL	\696
\699-CC	CC20:CC20A	\696
\70	DIP14	\70
\7022	DIP16:DIP16\SO	\7022
\7046	DIP16:DIP16\SO	4046
\7074	DIP24	\7074
\7074-CC	CC28:CC28A	\7074
\72	DIP14	\72
\7266	DIP14:DIP14\SO	XNOR
\7266-CC	CC20:CC20A	XNOR
\73	DIP14:DIP14\SO	\73

Part Type	PADS 2000 Decal	PADS Logic Symbol
\74	DIP14: DIP14\SO	\74
\74-CC	CC20: CC20A	\74
\75	DIP16: DIP16\SO	\75
\756	DIP20	\756
\758	DIP14	\758
\759	DIP14	\758
\76	DIP16: DIP16\SO	\76
\760	DIP20: DIP20\SOL	\244
\762	DIP20	\762: \762A
\763	DIP20	\763: \763A
\77	DIP14	\77
\78	DIP14: DIP14\SO	\78
\800	DIP20	NAND4
\8003	DIP8	NAND2: NANDDM
\802	DIP20	NOR4
\804	DIP20	NAND2: NANDDM
\804-CC	CC20: CC20A	NAND2: NANDDM
\804A	DIP20	NAND2: NANDDM
\805	DIP20	NOR2: NORDM
\805-CC	CC20: CC20A	NOR2: NORDM
\805A	DIP20	NOR2: NORDM
\808	DIP20	AND2
\808-CC	CC20: CC20A	AND2
\808A	DIP20	AND2

Part Type	PADS 2000 Decal	PADS Logic Symbol
\821	DIP24: DIP24\SOL: DIP24\300	821
\821-CC	CC28: CC28A	\821
\822	DIP24: DIP24\SOL: DIP24\300	\822
\822-CC	CC28: CC28A	\822
\823	DIP24: DIP24\SOL: DIP24\300	\823
\823-CC	CC28: CC28A	\823
\824	DIP24: DIP24\SOL: DIP24\300	\824
\824-CC	CC28: CC28A	\824
\825	DIP24: DIP24\SOL: DIP24\300	\825
\825-CC	CC28: CC28A	\825
\826	DIP24: DIP24\SOL: DIP24\300	\826
\826-CC	CC28: CC28A	\826
\832	DIP20	OR2
\832-CC	CC20: CC20A	OR2
\832A	DIP20	OR2
\83A	DIP16: DIP16\SO	\83A
\841	DIP24: DIP24\SOL: DIP24\300	\841
\841-CC	CC28: CC28A	\841
\842	DIP24: DIP24\SOL: DIP24\300	\842
\842-CC	CC28: CC28A	\842
\843	DIP24: DIP24\SOL: DIP24\300	\843
\844	DIP24: DIP24\SOL: DIP24\300	\843
\845	DIP24: DIP24\SOL: DIP24\300	\845
\845-CC	CC28: CC28A	\845

Part Type	PADS 2000 Decal	PADS Logic Symbol
\846	DIP24:DIP24\SOL:DIP24\300	\846
\846-CC	CC28:CC28A	\846
\85	DIP16:DIP16\SO	\85
\85-CC	CC20:CC20A	\85
\850	DIP28\600	\850
\850-CC	CC28:CC28A	\850
\851	DIP28\600	\851
\851-CC	CC28:CC28A	\851
\852	DIP24	\852
\852-CC	CC28:CC28A	\852
\856	DIP24	\856
\856-CC	CC28:CC28A	\856
\857	DIP24	\857
\86	DIP14:DIP14\SO	XOR
\86-CC	CC20:CC20A	XOR
\866	DIP28\600	\866
\867	DIP24	\867
\867-CC	CC28:CC28A	\867
\869	DIP24	\867
\869-CC	CC28:CC28A	\867
\870	DIP24	\870
\870-CC	CC28:CC28A	\870
\871	DIP28\600	\871
\871-CC	CC28:CC28A	\871

Part Type	PADS 2000 Decal	PADS Logic Symbol
\873	DIP24	\873
\874	DIP24	\874
\876	DIP24	\876
\877	DIP24	\877
\877-CC	CC28:CC28A	\877
\878	DIP24	\874
\879	DIP24	\876
\880	DIP24	\880
\881	DIP24:DIP24\SOL	\181
\881-CC	CC28:CC28A	\181
\881A	DIP24:DIP24\SOL	\181
\881A-CC	CC28:CC28A	\181
\882	DIP24:DIP24\SOL:DIP24\300	\882
\882A	DIP24:DIP24\SOL:DIP24\300	\882
\882A-CC	CC28:CC28A	\882
\885	DIP24	\885
\885-CC	CC28:CC28A	\885
\89	DIP16	\89
\90	DIP14	\90
\90A	DIP14	\90
\91	DIP14:DIP14\SO	\91
\91A	DIP14:DIP14\SO	\91
\92	DIP14:DIP14\SO	\92
\92A	DIP14:DIP14\SO	\92

Part Type	PADS 2000 Decal	PADS Logic Symbol
\93	DIP14:DIP14\SO	\92
\94	DIP16	\94
\95	DIP14:DIP14\SO	\95
\95A	DIP14:DIP14\SO	\95
\96	DIP16	\96
\963	DIP20:DIP20\SOL	\963
\963-CC	CC20:CC20A	\963
\964	DIP20:DIP20\SOL	\964
\964-CC	DIP20:DIP20\SOL	\964
\97	DIP16	\97
\990	DIP20:DIP20\SOL	\990
\990-CC	CC20:CC20A	\990
\991	DIP20:DIP20\SOL	\991
\991-CC	CC20:CC20A	\991
\992	DIP24:DIP24\SOL	\992
\993	DIP24:DIP24\SOL	\993
\993-CC	CC28:CC28A	\993 \993-DW DIP24\SOL
\994	DIP24:DIP24\SOL	\994
\994-CC	CC28:CC28A	\994
\995	DIP24:DIP24\SOL	\995
\996	DIP24:DIP24\SOL	\996
\996-CC	CC28:CC28A	\996

PART II

PCB DECALS

Part II of this Appendix contains 21 Drawing Sheets showing plots of PCB Decals. An index of these Decals is as follows:

INDEX

DECAL	PAGE NO.
CAPACITORS	1,2
CHIP CARRIERS	4
CONVENTIONAL DIPS	10, 11, 12
CRYSTALS, MOIRES AND MOUNTING HOLES	20
DECOUPLING CAPS	8
DIODES	9
EDGE FINGERS	13
GATE ARRAYS	5, 6, 7
HEADERS AND CONNECTORS	3
RESISTORS	15
SIPS	16, 17
SURFACE MOUNT DIPS	14
TRANSISTORS	18, 19

Naming Conventions

The following conventions have been used (for the most part) to give names to the Part Decals and Part Types.

Part Decal Naming Convention

Ceramic leadless chip carriers - CL

CL20 CL28 CL44 CL52 CC40

Rectangular ceramic leadless chip carriers - CLR

CLR18 CLR20 CLR28 CLR32

Rectangular leaded chip carriers - CCR

CCR18 CCR32

Pin grid arrays - PGA

PGA\68PA PGA\68PB PGA\68PC PGA\84P

Dual inline package - DIP

DIP50\900 DIP52\900

Part Type Naming Conventions

If a part exists in a DIP version with N leads, and also exists in a ceramic leadless chip carrier with the same N leads, it is shown as the same part with a different DECAL.

ex: 2168 DIP20:CLR20

Here the 2168 exists in a 20 pin DIP (DIP20) and in a 20 pin rectangular ceramic leadless chip carrier (CLR20).

If a part exists in a DIP version with N leads, and also in a ceramic leadless chip carrier with M leads, it is coded as two parts each with their own DECAL.

ex: 27LS00 DIP16
and: 27LS00L CL20

Here the part exists in a 16 pin DIP (DIP16) and a 20 pin ceramic leadless chip carrier (CL20). The convention, which conforms to AMD's ordering scheme, is to postfix the letter L after the part code.

If a part exists in a DIP version with N leads, and in a ceramic leadless chip carrier with M leads and also in another ceramic leadless chip carrier with P leads, it is shown as three parts with their own DECAL.

ex: 27LS31 DIP24\600
and: 27LS31L CLR32
and: 27LS31L-S CL28

Here the part exists in a 600 mil 24 pin DIP (DIP24\600), a 32 pin rectangular leadless chip carrier (CLR32) and also in a 38 pin ceramin leadless chip carrier (CL28). Postfixing the L and L-S conforms to AMD's ordering convention.

If a part exists in a DIP version with N leads and in a DIP version with M leads it is shown as two parts each with their own Decal.

ex: 10KP14 DIP20
and: 10KP14J DIP24\600

Here the part exists in both a 20 pin DIP (DIP20) and a 600 mil 24 pin DIP (DIP24\600). Postfixing the J conforms to AMD's ordering convention.

If a part exists in a LCC version with N leads and another LCC version with N leads it is shown as one part with 2 different Decals.

ex: 27S184L CLR28:CL28

Here it exists in both a square and a rectangular ceramic leadless chip carrier with identical pinouts.

Signetics Part Type Naming Conventions

If a part exists in multiple DIP versions, it is shown as different parts each conforming to the number of pinouts.

ex: 269N24 DIP24\400
and: 2692N28 DIP28\600
and: 269N40 DIP40\600

Here the part exists in a 400 mil 24 pin DIP (DIP24\400), a 600 mil 28 pin DIP (DIP28\600) and a 600 mil 40 pin DIP (DIP40\600). The naming convention used conforms to Signetics ordering codes.

If a part exists in a DIP, a PLCC (Plastic Leaded Chip Carrier), and a PGA (Pin Grid Array), it is shown as three parts each with its own Decal.

ex. 68010A CC68
and: 68010P PGA\68P
and: 68010 DIP64\900

Here the parts exists in a 900 mil 64 pin DIP (DIP64\900), a 68 pin leaded chip carrier (CC68), and a 68 pin PGA. The naming convention used conforms to Signetic ordering codes.

National Part Type Naming Conventions

If a part exists in multiple Decal versions, it is shown as different parts each with its own Decal.

ex: 16040 DIP48\600
and: 16040E PGA\68PB
and: 16040V CC68

Here the part exists in a 600 mil 48 pin DIP (DIP48\600), 68 pin leaded chip carrier, and a 68 pin PGA. This conforms to National's ordering convention.

Motorola Part Type Naming Conventions

If a part exists in multiple Decal versions, it is shown as different parts each with its own Decal.

ex: 146805E2 DIP40\600
and: 146805E27 CC40

Here the part exists in a 600 mil 40 pin DIP (DIP40\600), and a 40 pin leaded chip carrier with a different pinout. This conforms to Motorola's ordering convention.

Intel Part Type Naming Conventions

If a part exists in multiple Decal versions, it is shown as different parts, each with its own Decal.

ex: 8031 DIP40\600
and: N8031 CC48

Here the part exists in a 600 mil 40 pin DIP (DIP40\600), and a 48 pin leaded chip carrier with a different pinout. This conforms to Intel's ordering convention.

Texas Instruments Part Type Naming Conventions

If a part exists in a DIP version and chip carrier version, it is coded as two parts each with its own Decal.

ex: 7400 DIP14
and: 7400\CC DIP24\600

If a part exists in two DIP versions it is coded as two parts, each with its own Decals.

ex: 741181\300 DIP24
741181\600 DIP24\600

Power and Ground Naming Conventions

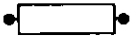
VCC	+5V
VEE	-5V
VDD	+12V
GND	Ground
AGND	Analog Ground
VPL9	+9V
VMN	-9V

CAP\MA10

CAP\MA20

CAP\MA30

CAP\MA40



CAP\MA50

CAP\MA60

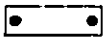
DRAWN BY : MMH	DATE 11-09-87	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON MA. 01460	
CHECKED BY:	DATE		
APPROVED BY:	DATE	TITLE CAPACITORS	
REVISION: A	DATE 11-09-87		
JOB NAME: CAPS1.JOB		REV. A	SCALE 1 : 1 | SHEET 1 OF 21

CAP\MR04

CAP\MR05

CAP\MR06

CAP\MR07

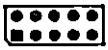


CAP\MR08

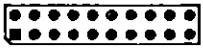
CAP\CR08

DRAWN BY : MMH	DATE 11-09-87	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON MA. 01460			
CHECKED BY:	DATE				
APPROVED BY:	DATE	TITLE CAPACITORS			
REVISION: A	DATE 11-09-87				
JOB NAME: CAPS2.JOB		REV. A	SCALE 1. : 1	SHEET 2 OF 21	

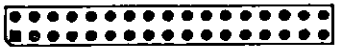
HEADER10



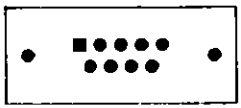
HEADER20



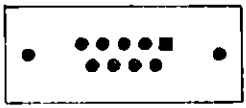
HEADER34



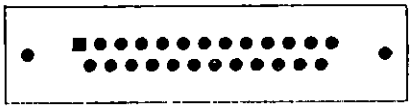
DCON9M



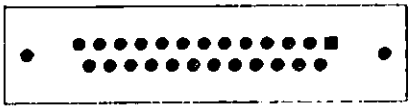
DCON9F



DCON25M



DCON25F

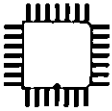


DRAWN BY : MMH	DATE 11-09-87	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON MA. 01460	
CHECKED BY:	DATE		
APPROVED BY:	DATE	TITLE HEADERS AND CONNECTORS	
REVISION: A	DATE 11-09-87		
JOB NAME: CONN1.JOB		REV. A	SCALE 1 : 1	SHEET 3 OF 21

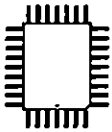
CC20

CCR18

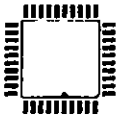
CC28



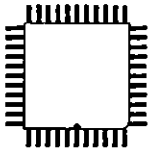
CCR32



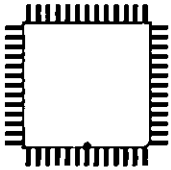
CC40\40



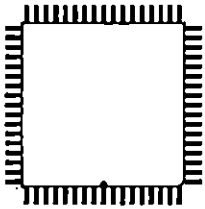
CC44



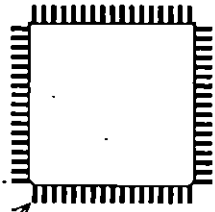
CC52



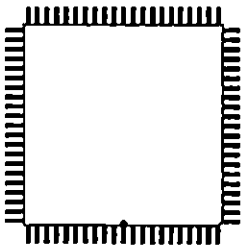
CC68



CC68A

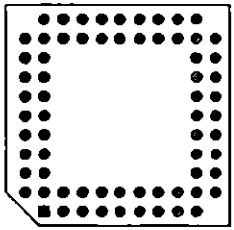


CC84

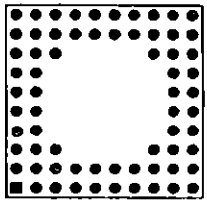


DRAWN BY : MMH	DATE 11-09-87	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON MA. 01460			
CHECKED BY:	DATE				
APPROVED BY:	DATE	TITLE CHIP CARRIERS			
REVISION: A	DATE 11-09-87				
JOB NAME: CHPCRS.JOB		REV. A	SCALE 1 : 1	SHEET 4 OF 21	

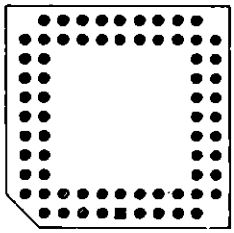
PGA\68P



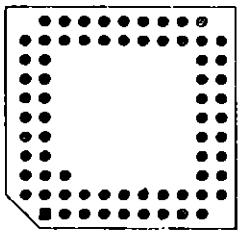
PGA\68PA



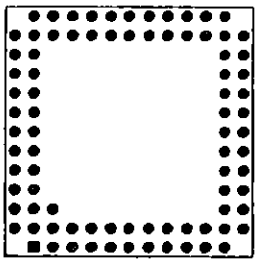
PGA\68PB



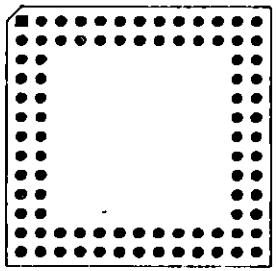
PGA\68PC



PGA\84P

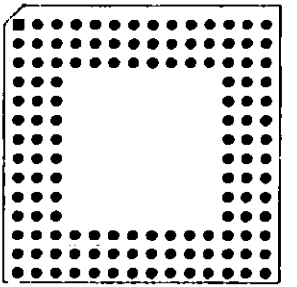


PGA\88P

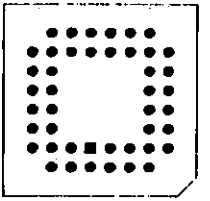


DRAWN BY : MMH	DATE 11-06-87	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON MA. 01460			
CHECKED BY:	DATE				
APPROVED BY:	DATE	TITLE GATE ARRAYS			
REVISION: A	DATE 11-06-87				
JOB NAME: GTARRAY1.JOB		REV. A	SCALE 1 : 1	SHEET 5 OF 21	

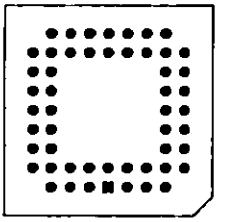
PGA\132P



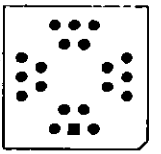
CL44



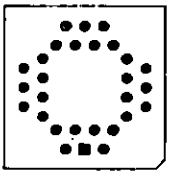
CL52



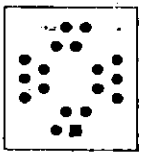
CL20



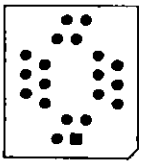
CL28



CLR18

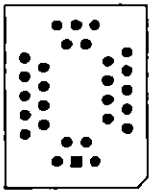


CLR20

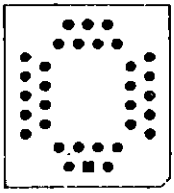


DRAWN BY : MHH	DATE 11-09-87	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON MA. 01460			
CHECKED BY:	DATE				
APPROVED BY:	DATE	TITLE GATE ARRAYS			
REVISION: A	DATE 11-09-87				
JOB NAME: GTARRAY2.JOB		REV. A	SCALE 1 : 1	SHEET 6 OF 21	

CLR28



CLR32



DRAWN BY : MWH	DATE 11-10-87	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON MA. 01460	
CHECKED BY:	DATE		
APPROVED BY:	DATE	TITLE GATE ARRAYS	
REVISION: A	DATE 11-10-87		
JOB NAME: GTARRAY3.JOB		REV. A	SCALE 1 : 1	SHEET 7 OF 21

DCAP1

DCAP\SR15

DCAP\SR20

DCAP\SR21

DCAP\SR26

DCAP\SR27

DRAWN BY : MMH	DATE 10-28-87	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON MA. 01460	
CHECKED BY:	DATE		
APPROVED BY:	DATE	TITLE DECOUPLING CAPS	
REVISION: A	DATE 10-28-87		
JOB NAME: DECAP.JOB		REV. A	SCALE 1 : 1	SHEET 8 OF 21

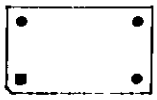
D07

D035

D041

DRAWN BY : MMH	DATE 11-06-87	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON MA. 01460	
CHECKED BY:	DATE		
APPROVED BY:	DATE	TITLE DIODES	
REVISION: A	DATE 11-06-87		
JOB NAME: DIODES.JOB		REV. A	SCALE 1 : 1	SHEET 9 OF 21

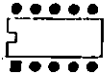
DI P4(14)



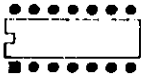
DI P6

DI P8

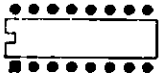
DI P10



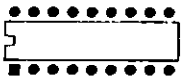
DI P14



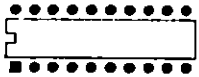
DI P16



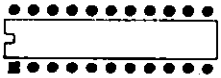
DI P18



DI P20

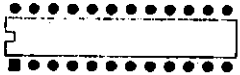


DI P22

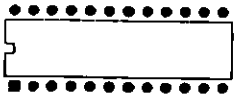


DI P22\400

DI P24

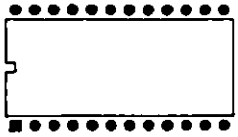


DI P24\400

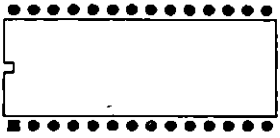


DRAWN BY : MMH	DATE 11-09-87	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON MA. 01460	
CHECKED BY:	DATE		
APPROVED BY	DATE	TITLE CONVENTIONAL DIPS	
REVISION: A	DATE 11-09-87		
JOB NAME: DIP1.JOB		REV. A	SCALE 1 : 1	SHEET 10 OF 21

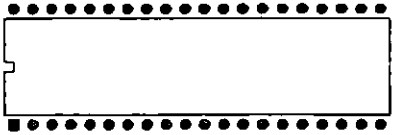
DIP24\600



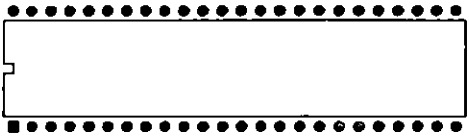
DIP28\600



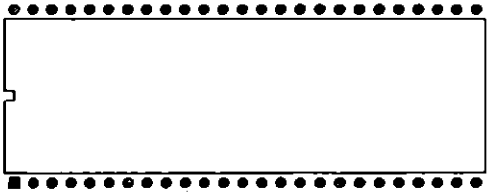
DIP40\600



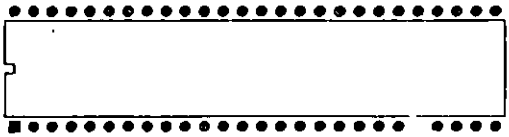
DIP48\600



DIP50\900

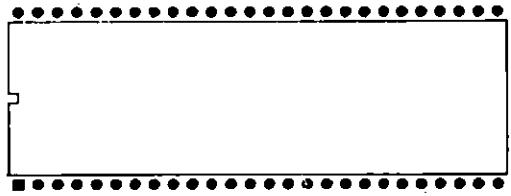


DIP52\600

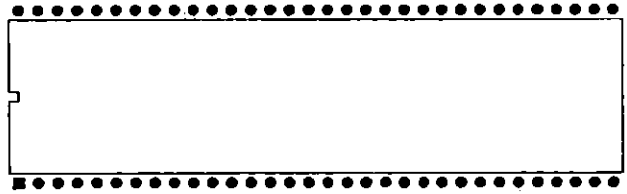


DRAWN BY : MMH	DATE 11-09-87	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON MA. 01460			
CHECKED BY:	DATE				
APPROVED BY:	DATE	TITLE CONVENTIONAL DIPS			
REVISION: A	DATE 11-09-87				
JOB NAME: DIP2.JOB		REV. A	SCALE 1 : 1	SHEET 11 OF 21	

DIP52\900



DIP64\900



DRAWN BY : MWH	DATE 11-09-87	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON MA. 01460	
CHECKED BY:	DATE		
APPROVED BY:	DATE	TITLE CONVENTIONAL DIPS	
REVISION: A	DATE 11-09-87		
JOB NAME: DIP3.JOB		REV. A	SCALE 1 : 1	SHEET 12 OF 21

CON\26P\ED

PIN 1

CON\60P\100\ED

PIN 1

CON\86P\156\ED

(Note: CON\86P\156\ED has been rotated 90 degrees counter clock wise for this drawing.)

PIN 1

DRAWN BY: MMH	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON MA. 01460	
CHECKED BY:		
APPROVED BY:	TITLE EDGE FINGERS	
DATE 11-06-87		
JOB NAME: EDGFNGR.JOB	REV. A	SCALE 1 : 1 SHEET 13 OF 21

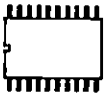
DI P8\SO

DI P14\SO

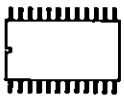
DI P16\SO

DI P16\SOL

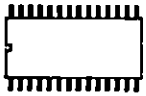
DI P20\SOL



DI P24\SOL



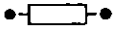
DI P28\SOL



DRAWN BY : MMH	DATE 11-05-87	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON MA. 01460			
CHECKED BY:	DATE				
APPROVED BY:	DATE	TITLE SURFACE MOUNT DIPS			
REVISION: A	DATE 11-05-87				
JOB NAME: SMDDIP1.JOB		REV. A	SCALE 1 : 1	SHEET 14 OF 21	

R1/8W

R1/4W



R1/4WB

R1/4WA

R1/2W

R1W

R2W

R3W

VRES

VRES\TOP\ADJ

DRAWN BY : MMH	DATE 11-09-87	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON MA. 01460	
CHECKED BY: TSH	DATE		
APPROVED BY: [illegible]	DATE	TITLE RESISTORS	
REVISION: A	DATE 11-09-87		
JOB NAME: RES1.JOB		REV. A	SCALE 1 : 1

SIP\1P

SIP\2P

SIP\3P

SIP\4P

SIP\5P

SIP\6P

SIP\7P

SIP\8P

SIP\9P

SIP\10P

SIP\11P

SIP\12P

DRAWN BY : MMH	DATE 11-06-87	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON MA. 01460			
CHECKED BY:	DATE				
APPROVED BY:	DATE	TITLE SIPS			
REVISION: A	DATE 11-06-87				
JOB NAME: SIP1.JOB		REV. A	SCALE 1 : 1	SHEET 16 OF 21	

SIP\13P

SIP\14P

SIP\15P

SIP\16P

SIP\17P

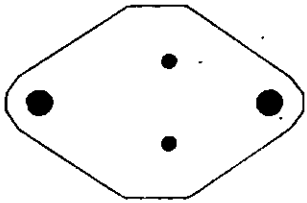
SIP\18P

SIP\19P

SIP\20P

DRAWN BY : MWH	DATE 11-06-87	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON MA. 01460			
CHECKED BY:	DATE				
APPROVED BY:	DATE	TITLE	SIPS		
REVISION: A	DATE 11-06-87				
JOB NAME: SIP2.JOB		REV. A	SCALE 1 : 1	SHEET 17 OF 21	

TO-3

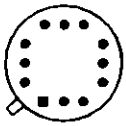


TO-5

TO-8

TO-8\200

TO-8\400



TO-18

TO-39

TO-52

TO-71

TO-72

DRAWN BY : MMH	DATE 10-27-87	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON MA. 01460			
CHECKED BY:	DATE				
APPROVED BY:	DATE	TITLE	TRANSISTORS		
REVISION: A	DATE 10-27-87				
JOB NAME: TRAN1.JOB		REV. A	SCALE 1 : 1	SHEET 18 OF 21	

TO-78

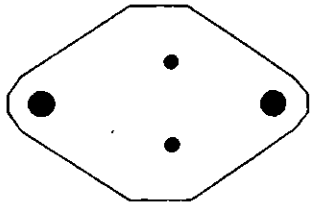
TO-92

TO-99

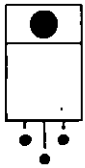
TO-100

TO-100\210

TO-204



TO-220

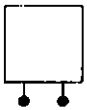


TO-220\UP

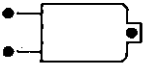
TO-220\UP\4P

DRAWN BY : MMH	DATE 10-27-87	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON MA. 01460			
CHECKED BY: DW	DATE				
APPROVED BY:	DATE	TITLE TRANSISTORS			
REVISION: A	DATE 10-27-87				
JOB NAME: TRAN2.JOB		REV. A	SCALE 1 : 1	SHEET 19 OF 21	

XTAL1



XTAL2



MOIRE

MTHOLE1

MTHOLE2

DRAWN BY : MWH	DATE 11-09-87	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON MA. 01460			
CHECKED BY:	DATE				
APPROVED BY:	DATE	TITLE CRYSTALS, MOIRE, AND MTG. HOLES			
REVISION: A	DATE 11-09-87				
JOB NAME: CRYSMTG.JOB		REV. A	SCALE 1 : 1	SHEET 20 OF 21	

DRAWN BY :	DATE	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON MA. 01460		
CHECKED BY:	DATE			
APPROVED BY:	DATE	TITLE *******		
REVISION:	DATE			
JOB NAME:		REV.	SCALE :	SHEET 21 OF 21

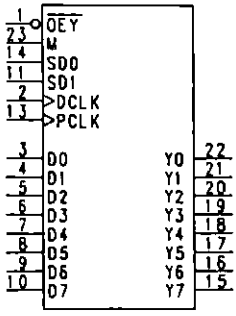
PART III

CIRCUIT SYMBOL PLOTS

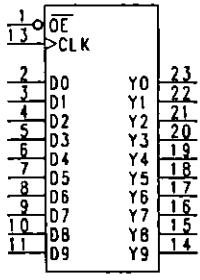
Part III of this Appendix contains some 113 Drawing Sheets showing plots of PADS-LOGIC Schematic Symbols. Each drawing sheet contains a number of Schematic Symbol Plots. An Index of these Symbols is as follows:

INDEX

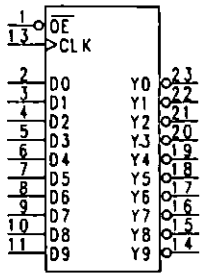
SYMBOLS	PAGE NO.
AM29818 THROUGH AM29864	1, 2
CAPS	3
DIPS	4-8
GATES	9-11
GENERIC DIPS	12-16
MISC. COMPONENTS	82-83
OP AMPS	17
PALS	18-23
PINS	26
POWER DEVICES	27
R* (PROMS/RAMS)	28-33
RELAYS	34
RESISTORS	35
TXXXXXX	36
1XXXXXX	37-40
2XXXXXX	41-47
3XXXXXX	48-52
4XXXXXX	53-57
6XXXXXX	58-63
7XXXXXX	64
8XXXXXX	65-81
9XXXXXX	82-84
TTL 54/74 SERIES \107 - \996	85-113



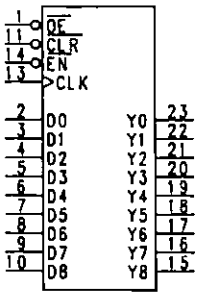
29818



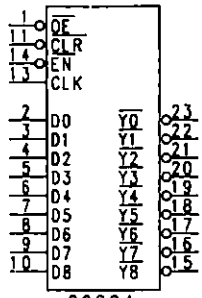
29821



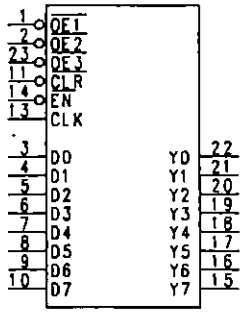
29822



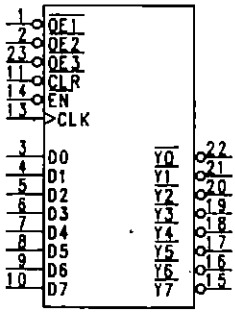
29823



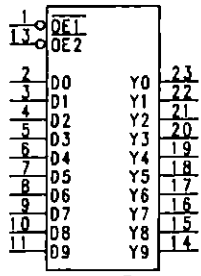
29824



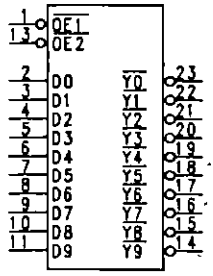
29825



29826

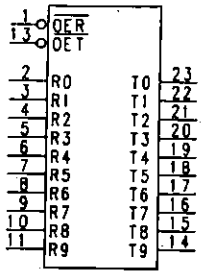


29827

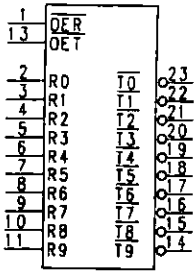


29828

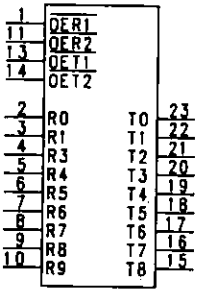
DRAWN BY: S.P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460	
CHECKED BY:	DATE 1-3-90		
APPROVED BY:	DATE 1-3-90	TITLE AM298**	
REVISION:	DATE		
JOB NAME:	REV. A	SCALE 1:1	SHEET 1 OF 113



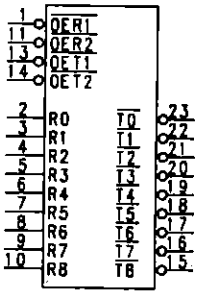
29861



29862

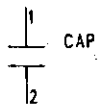


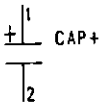
29863

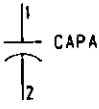


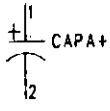
29864

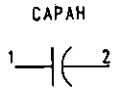
DRAWN BY: JP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: TP	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE AM298**		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 2 OF 113

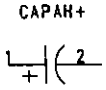


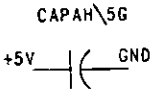


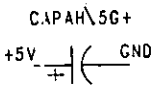


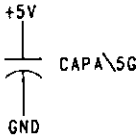


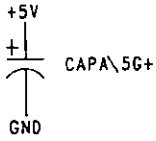


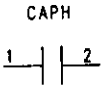


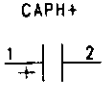


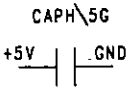


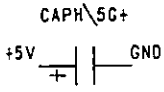


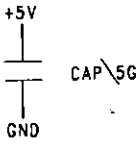


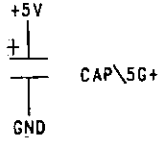




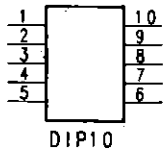




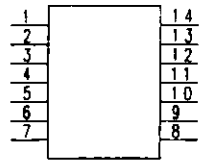




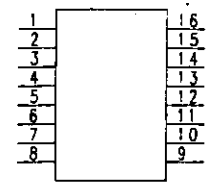
DRAWN BY: JP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: [illegible]	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE CAPS		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 3 OF 113



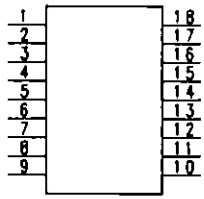
DIP10



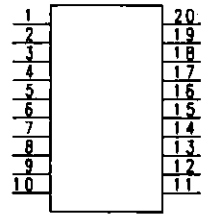
DIP14



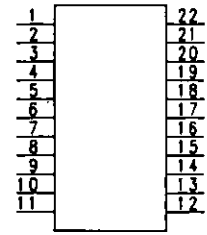
DIP16



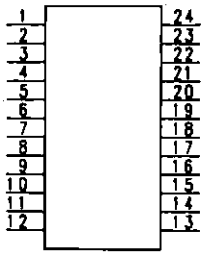
DIP18



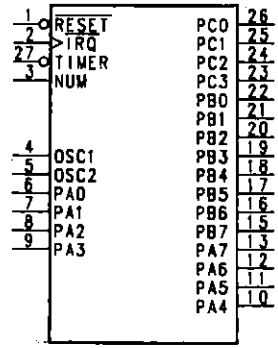
DIP20



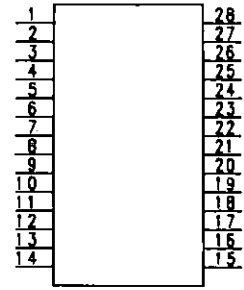
DIP22



DIP24

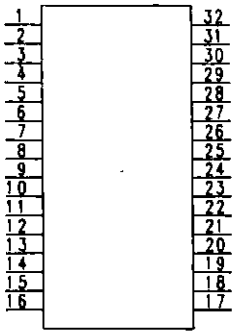


DIP26

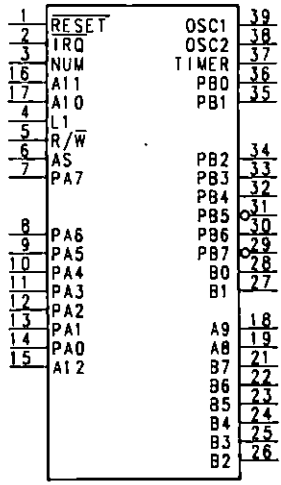


DIP28

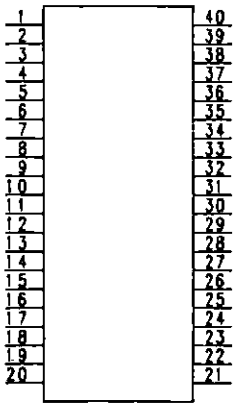
DRAWN BY: [illegible]	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: [illegible]	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE DIP		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 4 OF 113



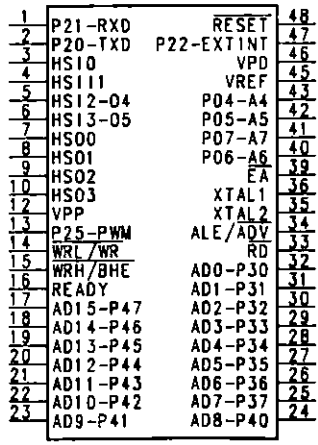
DIP32



DIP38

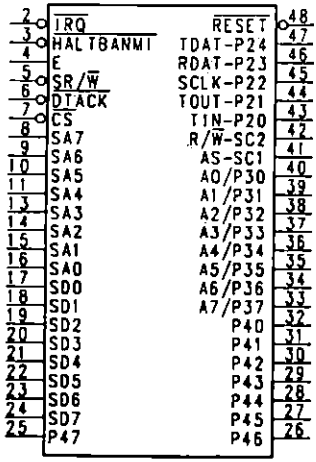


DIP40

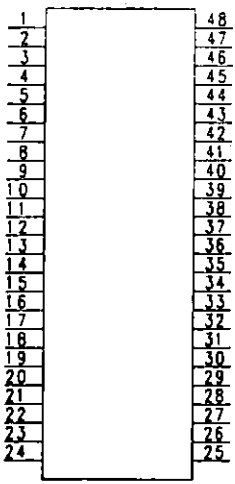


DIP44

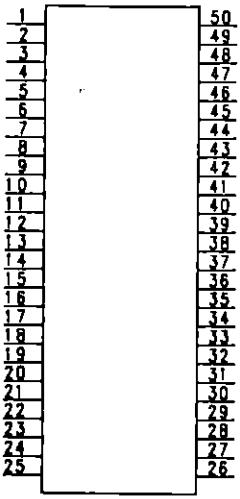
DRAWN BY: J.P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE DIP		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 5 OF 113



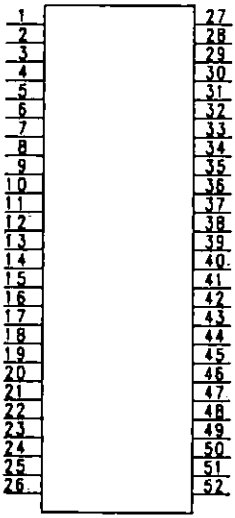
DIP46



DIP48

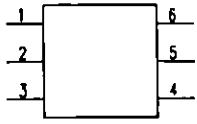


DIP50

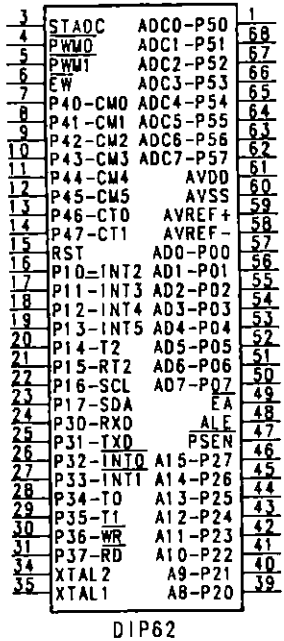


DIP52

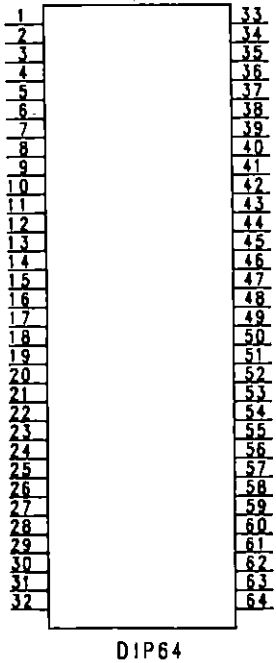
DRAWN BY: L.P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460	
CHECKED BY: [illegible]	DATE 1-3-90		
APPROVED BY: [illegible]	DATE 1-3-90	TITLE DIP	
REVISION:	DATE		
JOB NAME:	REV. A	SCALE 1:1	SHEET 6 OF 113



DIP6

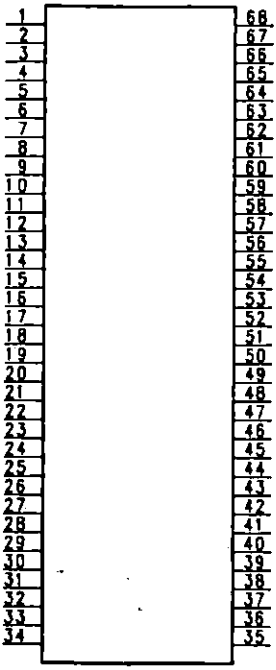


DIP62

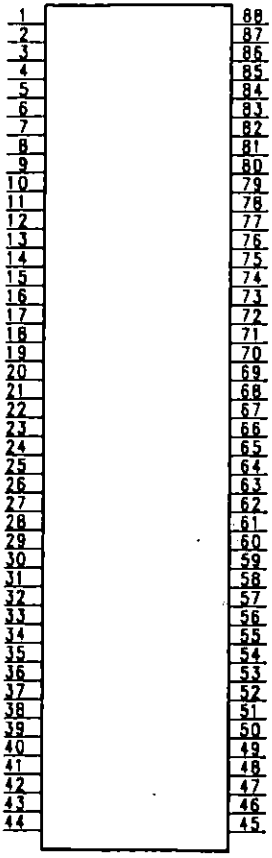


DIP64

DRAWN BY: J. P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE DIP		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 7 OF 113

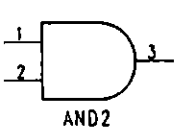


DIP8

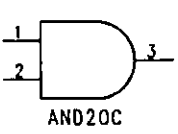


DIP88

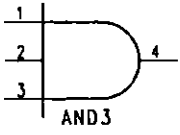
DRAWN BY: J.P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460	
CHECKED BY:	DATE 1-3-90		
APPROVED BY:	DATE	TITLE DIP	
REVISION:	DATE		
JOB NAME:		REV. A	SCALE 1:1	SHEET 8 OF 113



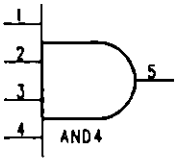
AND2



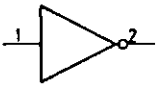
AND2OC



AND3

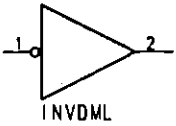


AND4

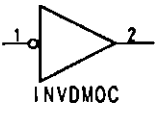


INV

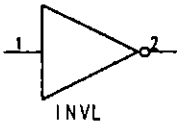
INVDM



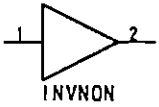
INVDML



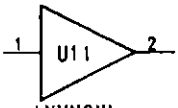
INVDMOC



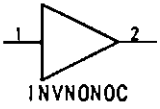
INVL



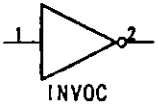
INVNON



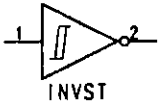
INVNONL



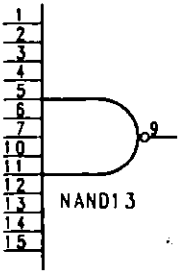
INVNONOC



INVOC



INVST



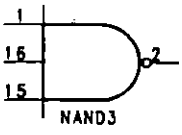
NAND13

DRAWN BY: P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3 90			
APPROVED BY:	DATE 1-3-90	TITLE GATES		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 9 OF 113

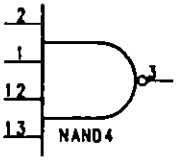
NAND2

NAND2DM

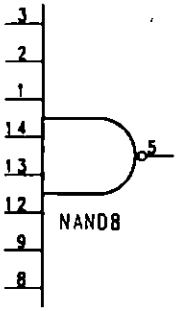
NAND2OC



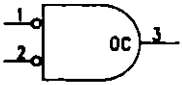
NAND3



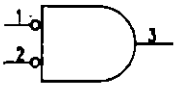
NAND4



NAND8



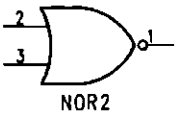
NANDDMOC



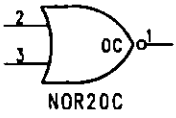
NANDDM

NANDST

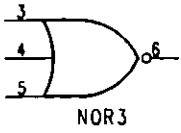
DRAWN BY: J.P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: DC	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE GATES		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 10 OF 113



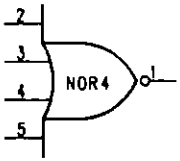
NOR2

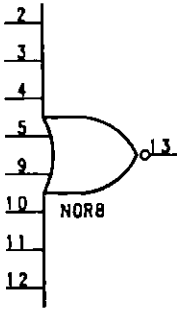


NOR2OC

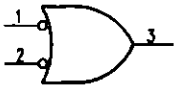


NOR3

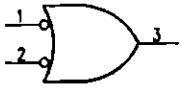




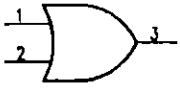
NOR8



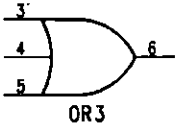
NORDM



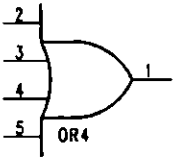
NORDMOC



OR2

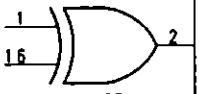


OR3

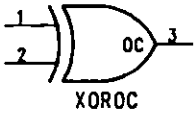


OR4

XNOR

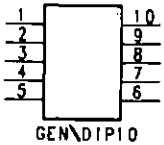


XOR

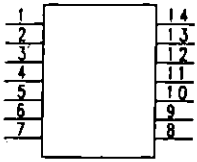


XOROC

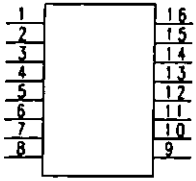
DRAWN BY: J. P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: [illegible]	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE GATES		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 11 OF 113



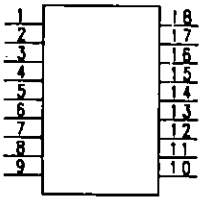
GEN\DIP10



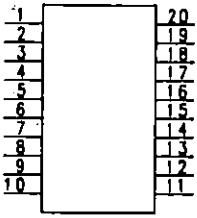
GEN\DIP14



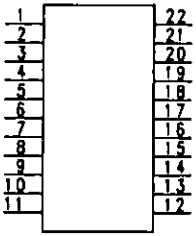
GEN\DIP16



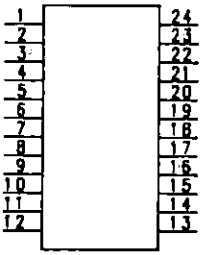
GEN\DIP18



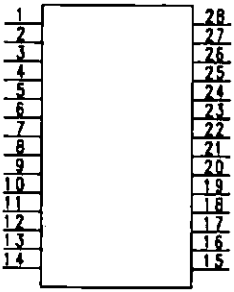
GEN\DIP20



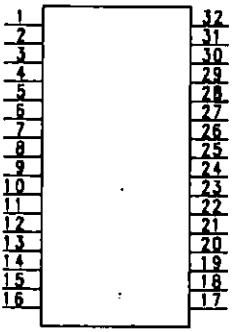
GEN\DIP22



GEN\DIP24

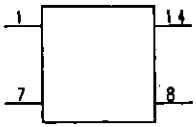


GEN\DIP28

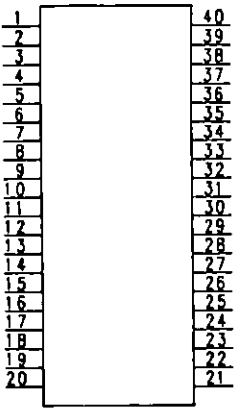


GEN\DIP32

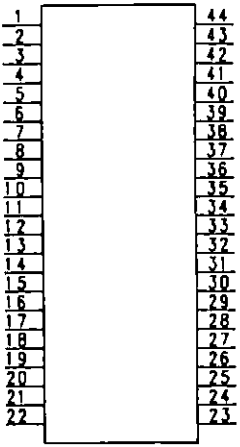
DRAWN BY: f P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: (illegible)	DATE 1-3-90			
APPROVED BY: (illegible)	DATE 1-3-90	TITLE GENERIC DIPS		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 12 OF 113



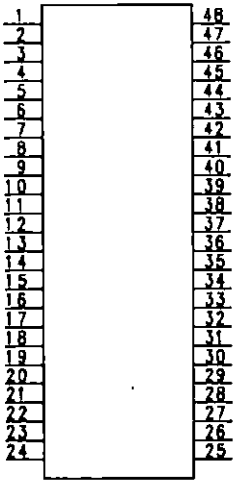
GEN\DIP4(14)



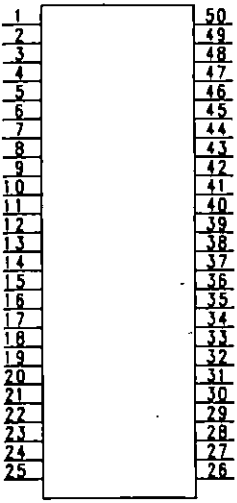
GEN\DIP40



GEN\DIP44

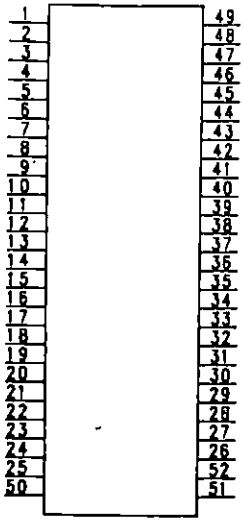


GEN\DIP48

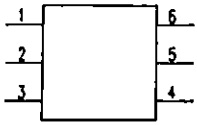


GEN\DIP50

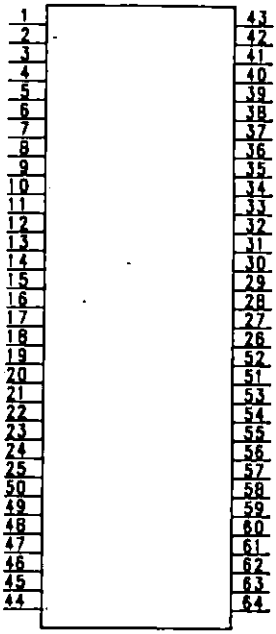
DRAWN BY: LP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE GENERIC DIPS		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 13 OF 113



GEN\DIP52

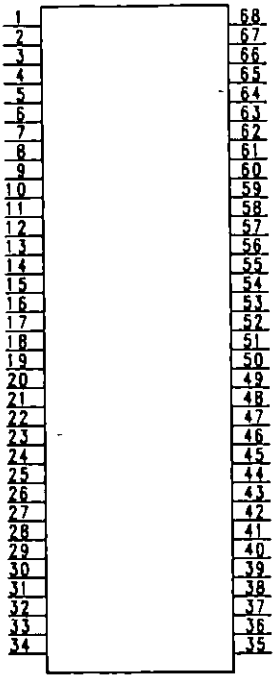


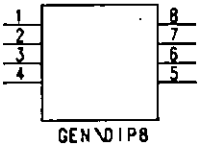
GEN\DIP6



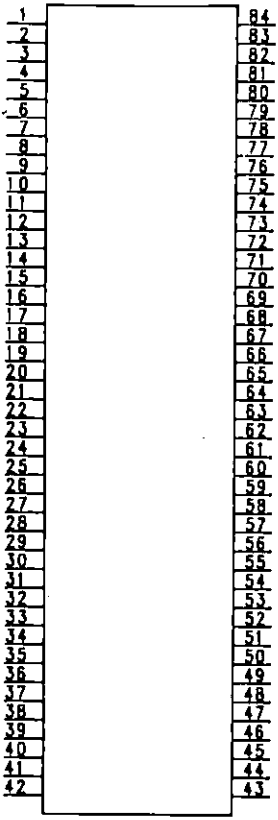
GEN\DIP64

DRAWN BY: J.P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: [illegible]	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE GENERIC DIPS		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 14 OF 113





DRAWN BY: LP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: JC	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE GENERIC DIPS		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 15 OF 113



GEN\DIP84

DRAWN BY: J.P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: [illegible]	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE GENERIC DIPS		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 16 OF 113

DIO-H

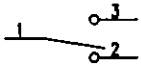
DIO-V

XTAL

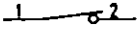
XTALH

EXTIN

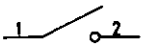
EXTOUT



SW-SPDT



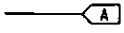
SW-SPST-NC



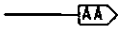
SW-SPST-NO

DRAWN BY: J.P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: [illegible]	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE DIODE, XTAL, IN/OUT AND SWITCHES		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 17 OF 113

OFF PAGE SYMBOLS



REFIN



REFOUT

GROUND SYMBOLS

GND

AGND

CHGND

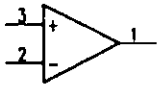
VOLTAGE SYMBOLS

+5V

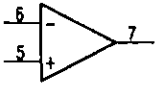
BUBBLE

Y

DRAWN BY: L.P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: [illegible]	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE OFFPAGE, POWER, AND GROUND		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 18 OF 113

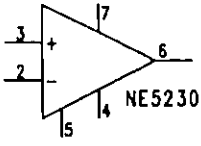


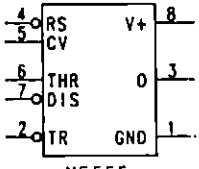
AOPAMP



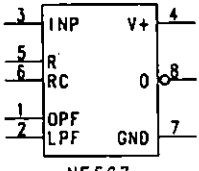
AOPAMPM

AOPAMPPN



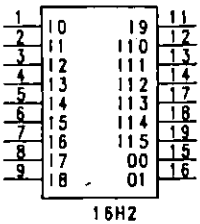


NE555

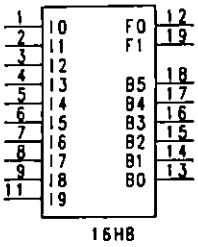


NE567

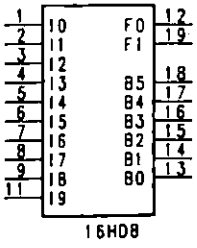
DRAWN BY: J.P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE OP AMPS		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 19 OF 113



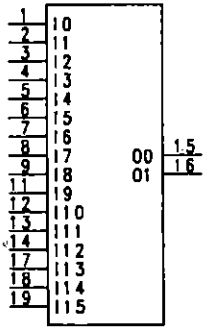
16H2



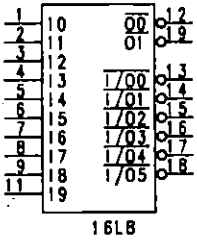
16H8



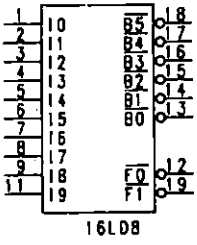
16HD8



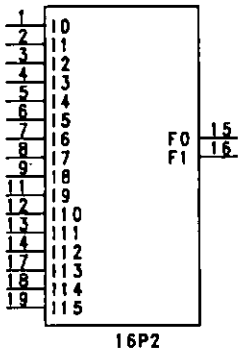
16L2



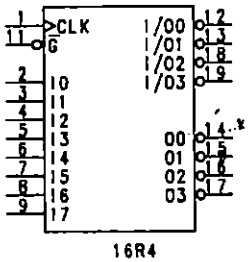
16LB



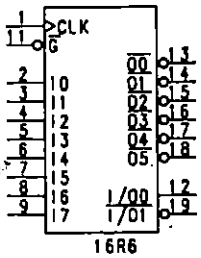
16LD8



16P2

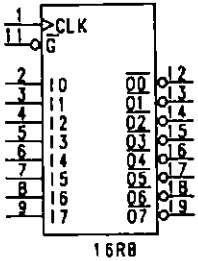


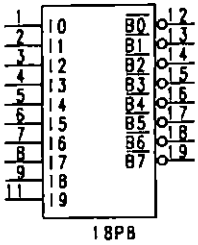
16R4

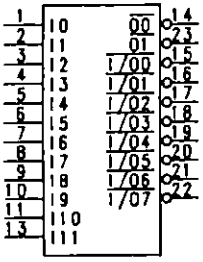


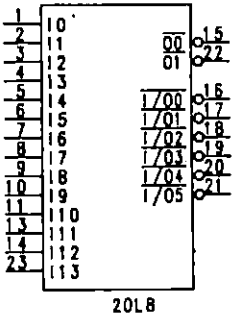
16R6

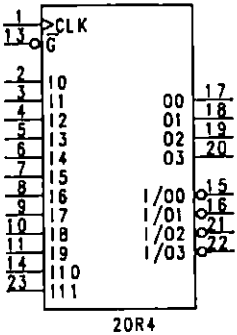
DRAWN BY: J.P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE PALS		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 20 OF 113

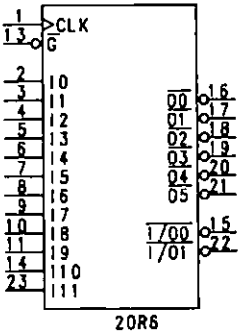


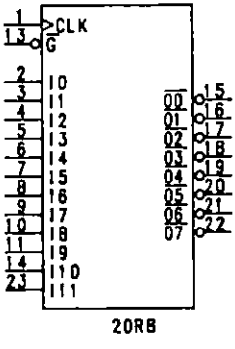


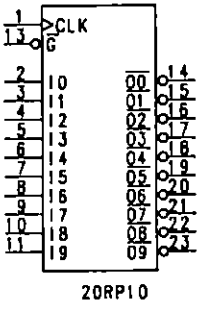


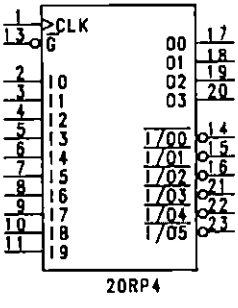




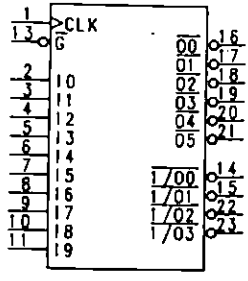




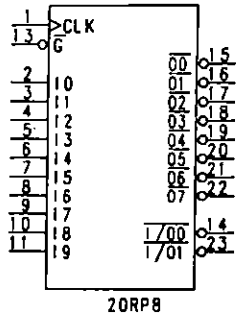




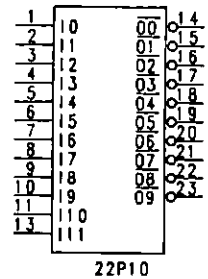
DRAWN BY: J.P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE PALS		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 21 OF 113



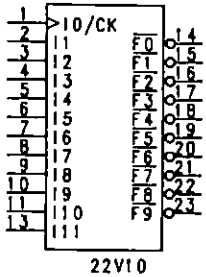
20RP6



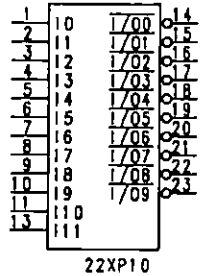
20RP8



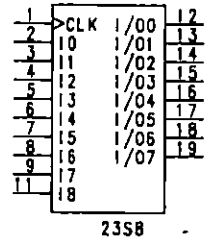
22P10



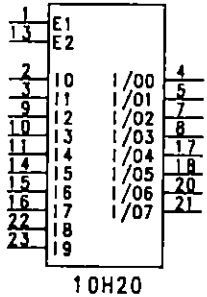
22V10



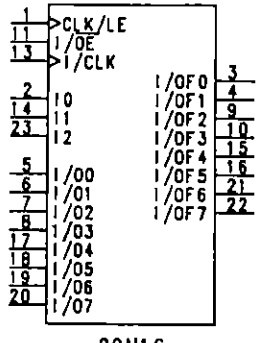
22XP10



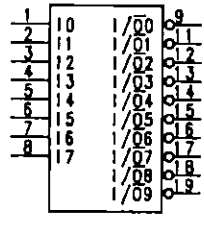
23S8



10H20

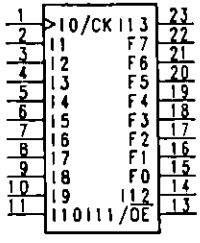


29M16

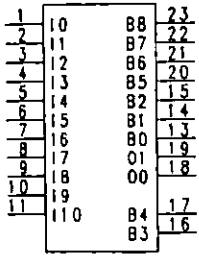


PLS153

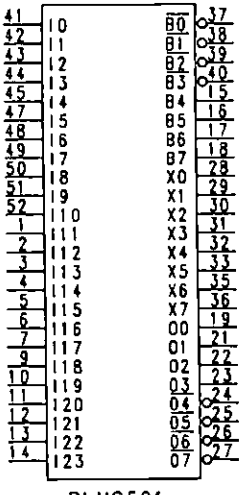
DRAWN BY: J. P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE PALS		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 22 OF 113



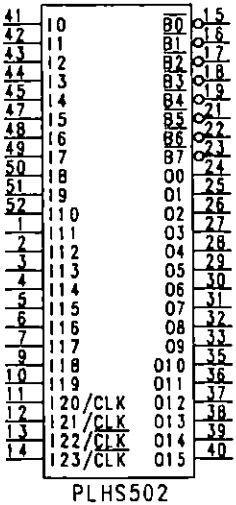
20V8



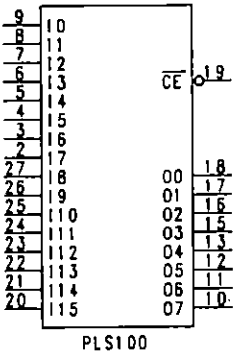
PLC473



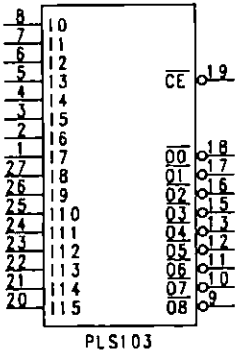
PLHS501



PLHS502

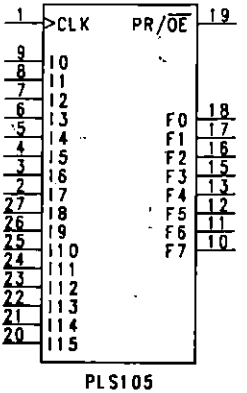


PLS100

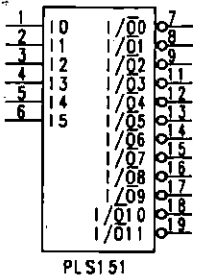


PLS103

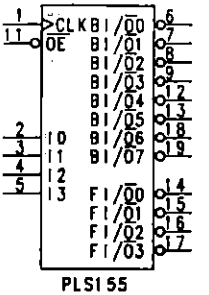
DRAWN BY: RP.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460	
CHECKED BY: [illegible]	DATE 1-3-90		
APPROVED BY: [illegible]	DATE 1-3-90	TITLE PALS	
REVISION:	DATE		
JOB NAME:	REV. A	SCALE 1:1	SHEET 23 OF 113



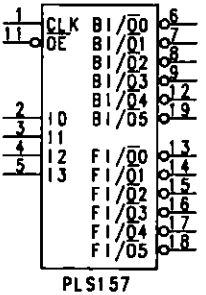
PLS105



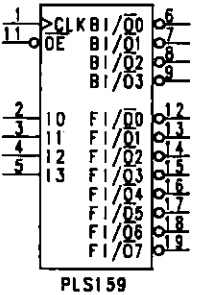
PLS151



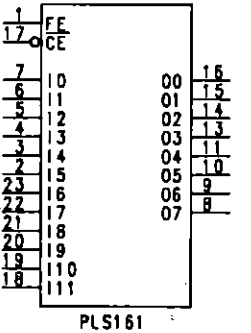
PLS155



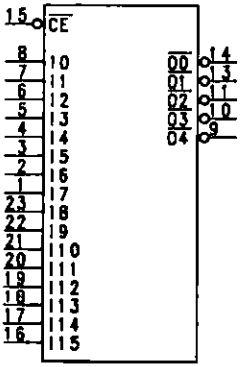
PLS157



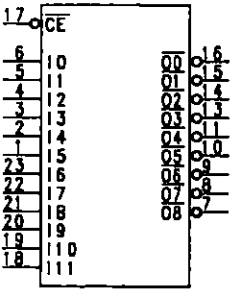
PLS159



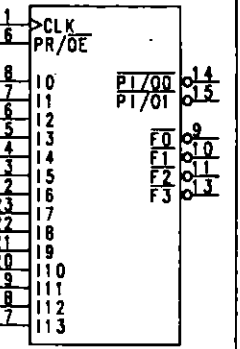
PLS161



PLS162

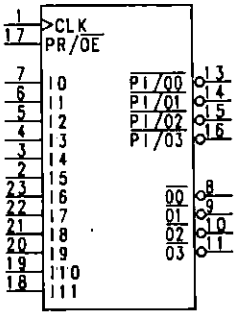


PLS163

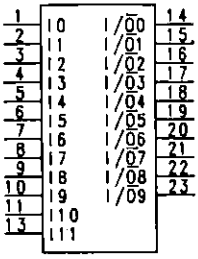


PLS167

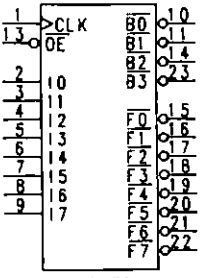
DRAWN BY: F.P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460	
CHECKED BY:	DATE 1-3-90		
APPROVED BY:	DATE 1-3-90	TITLE PALS	
REVISION:	DATE		
JOB NAME:	REV. A	SCALE 1:1	SHEET 24 OF 113



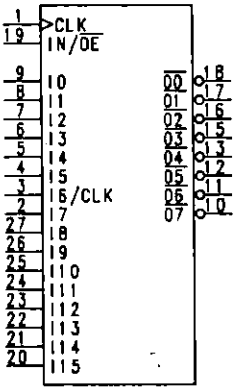
PLS168



PLS173



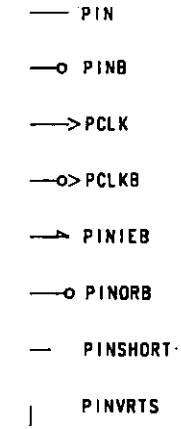
PLS179



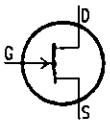
PLUS405

DRAWN BY: J.P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE PALS		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 25 OF 113

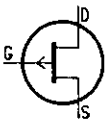
PINS



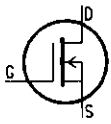
DRAWN BY: J. P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: DC	DATE 1-3-90			
APPROVED BY:	DATE	TITLE PINS		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 26 OF 113



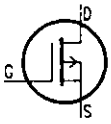
JFETN-V



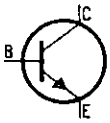
JFETP-V



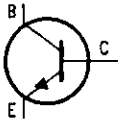
MFETN-V



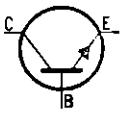
MFETP-V



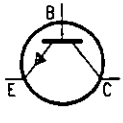
NPN-V



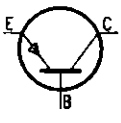
NPN-VA



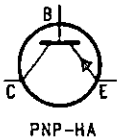
NPN-H



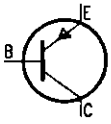
NPN-HA



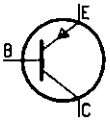
PNP-H



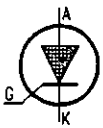
PNP-HA



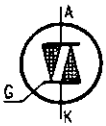
PNP-V



PNP-VA

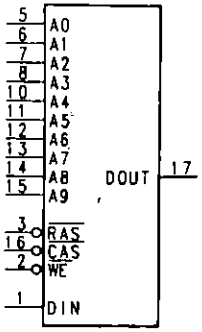


SCR-V

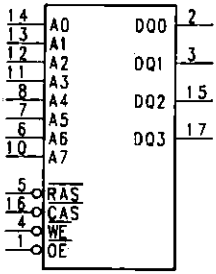


TRIAC-V

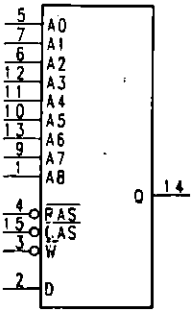
DRAWN BY: JP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: RD	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE POWER DEVICES		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 27 OF 113



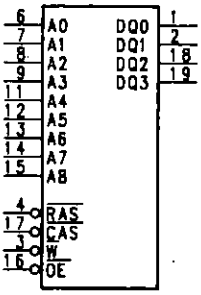
RD1024K1



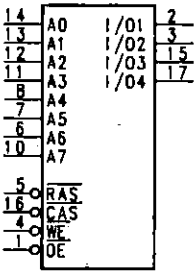
RD16K4



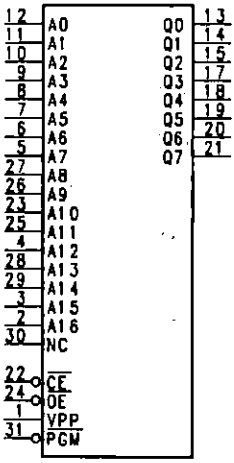
RD256K1



RD256K4

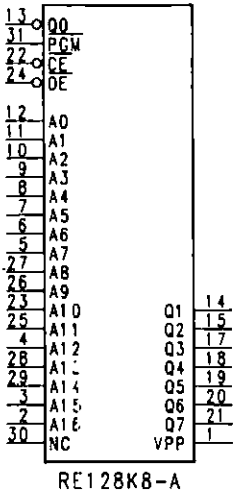


RD64K4

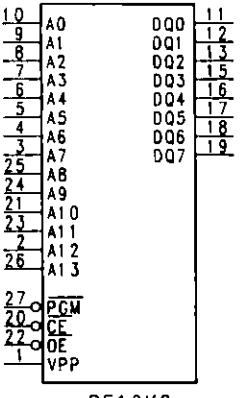


RE128K8

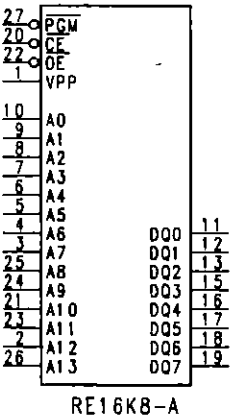
DRAWN BY: JP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: DC	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE R* (PROMS/RAMS)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 28 OF 113



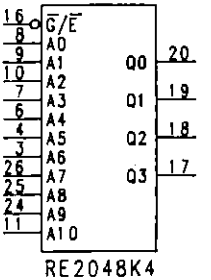
RE128K8-A



RE16K8

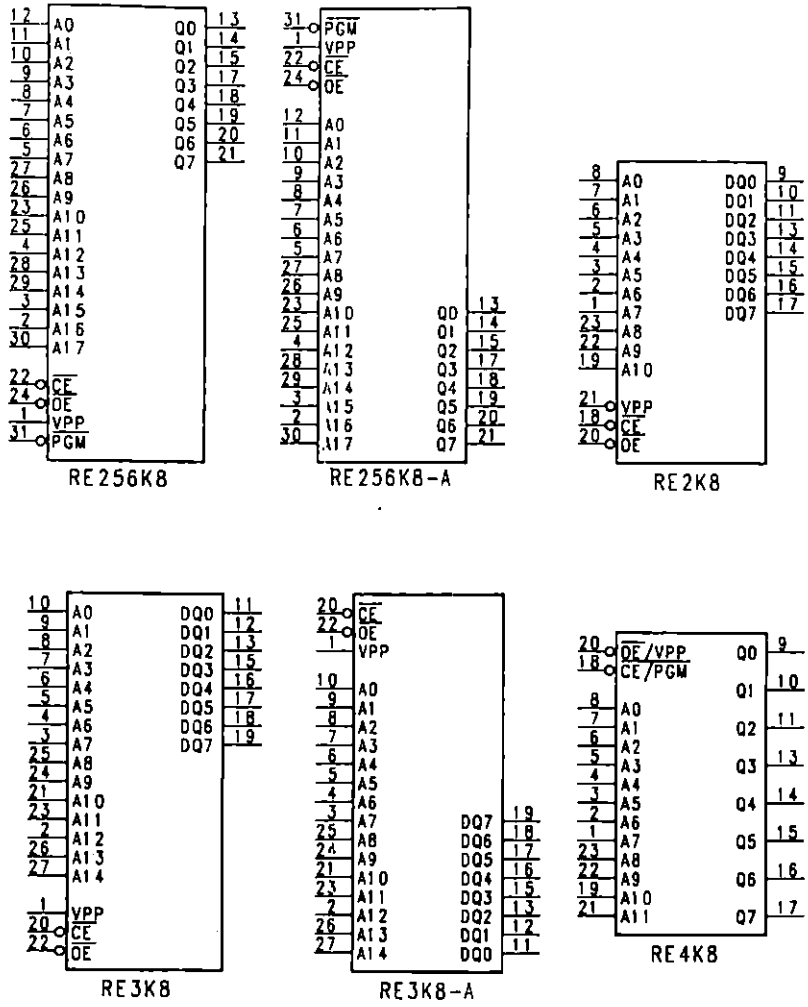


RE16K8-A

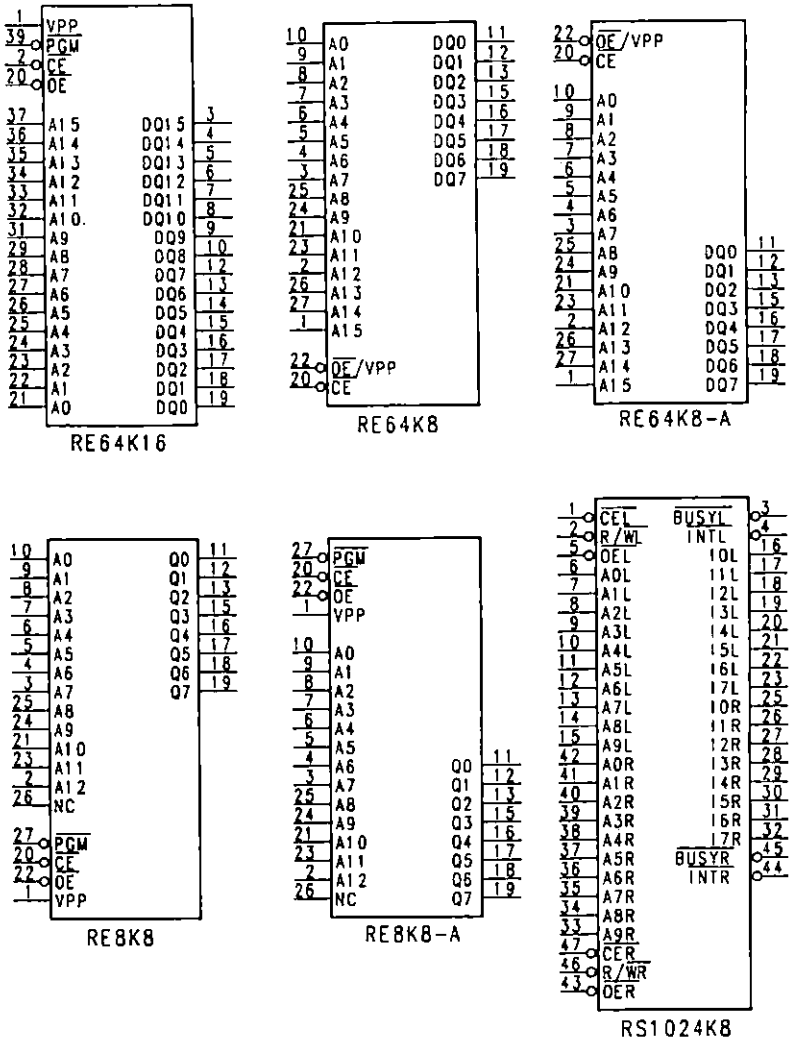


RE2048K4

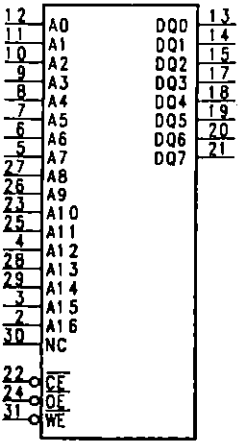
DRAWN BY: JP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: [illegible]	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE R* (PROMS/RAMS)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 29 OF 113



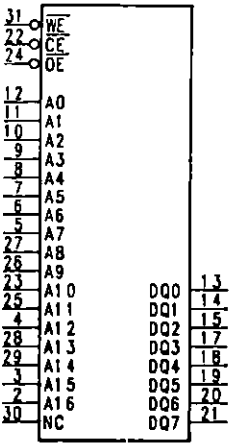
DRAWN BY: F.P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE R* (PROMS/RAMS)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 30 OF 113



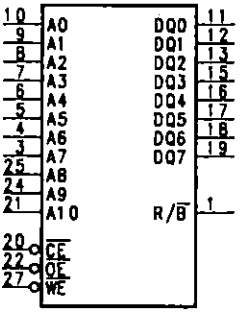
DRAWN BY: [illegible] P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: [illegible]	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE R* (PROMS/RAMS)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 31 OF 113



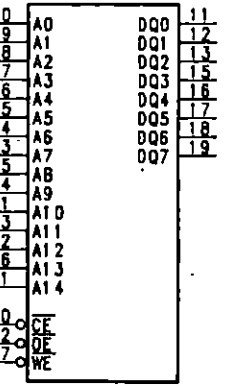
RS128KB



RS128K8-A

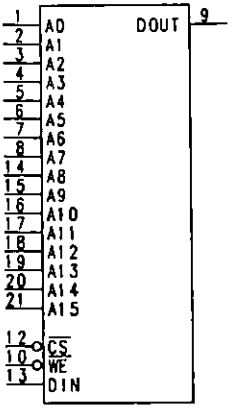


RS2K8

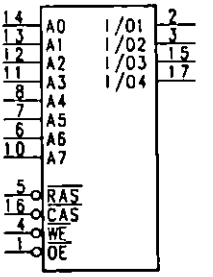


RS32K8

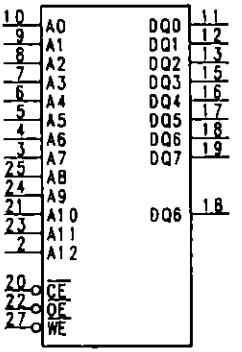
DRAWN BY: J P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: M	DATE 1-3-90			
APPROVED BY:	DATE	TITLE R* (PROMS/RAMS)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 32 OF 113



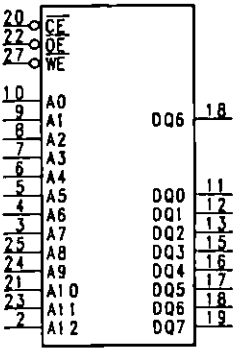
RS64K1



RS64K4

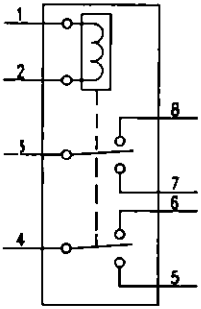


RS8K8

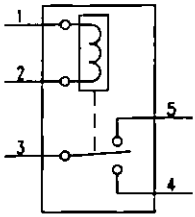


RS8K8-A

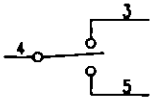
DRAWN BY: J.P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE	TITLE R* (PROMS/RAMS)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 33 OF 113



RLY-DPDT



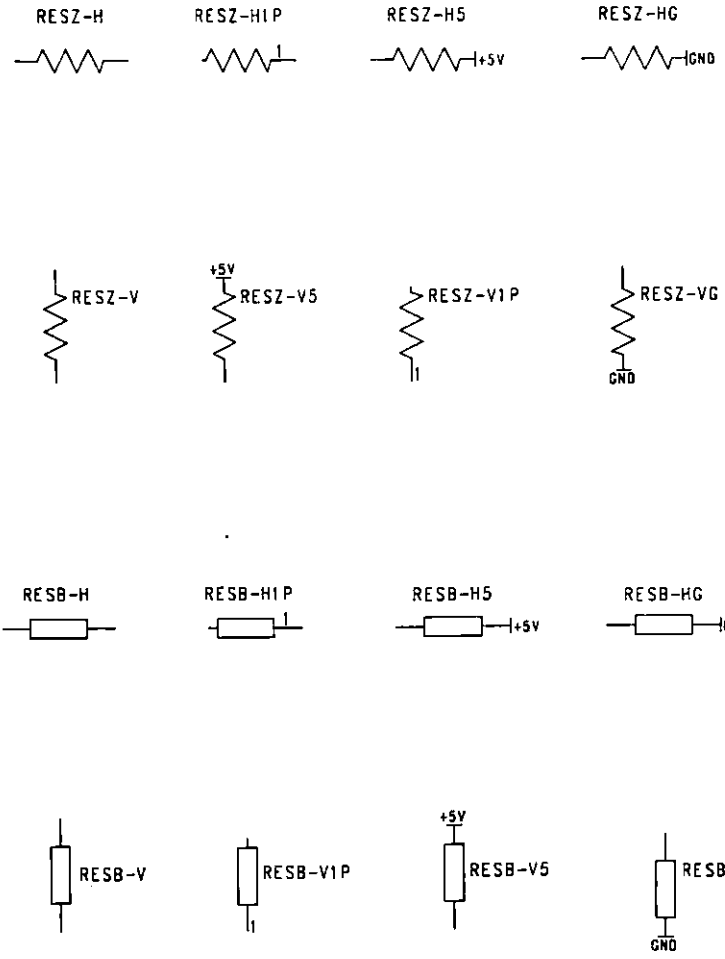
RLY-SPDT



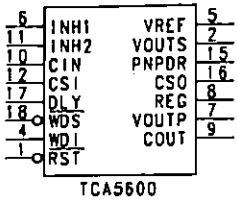
RLY-SW-SPDT

RLY-COIL

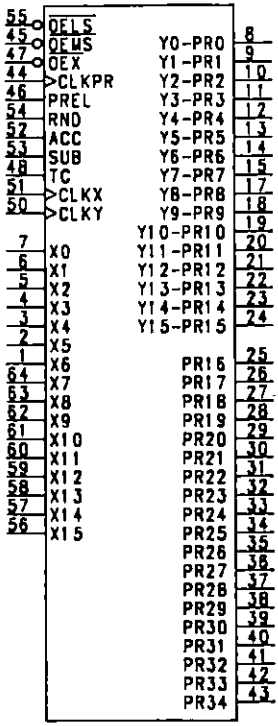
DRAWN BY: J.P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: [illegible]	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE RELAYS		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 34 OF 113



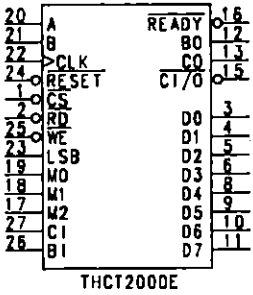
DRAWN BY: J P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: DC	DATE 1-3-90			
APPROVED BY:	DATE	TITLE RESISTORS		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 35 OF 113



TCA5600

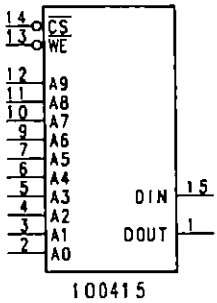


THCT1010

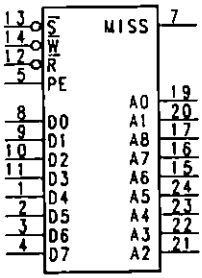


THCT2000E

DRAWN BY: LP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: DC	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE T******		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 36 OF 113

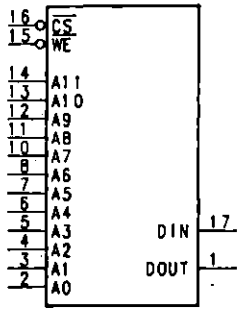


100415

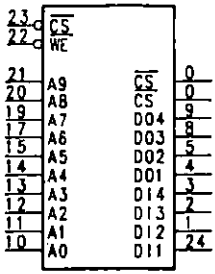


100469

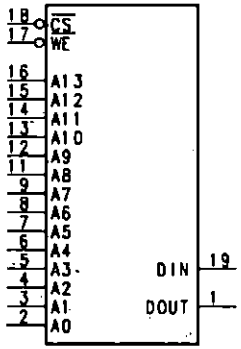
DRAWN BY: J P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460			
CHECKED BY:	DATE 1-3-90				
APPROVED BY:	DATE 1-3-90	TITLE 1XXXXXX			
REVISION:	DATE				
JOB NAME:		REV. A	SCALE 1:1	SHEET 37 OF 113	



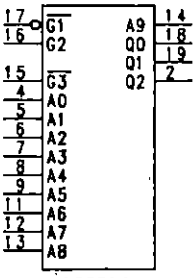
100470



100474

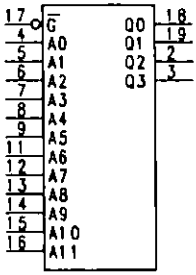


100480

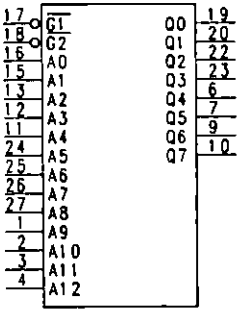


100P14

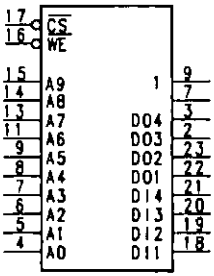
DRAWN BY: JP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: DC	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 1XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 38 OF 113



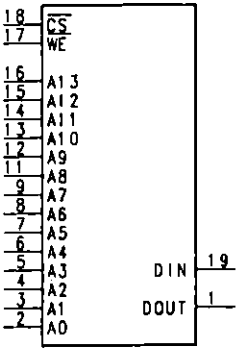
100P44



100P88

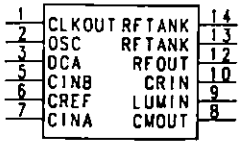


10474

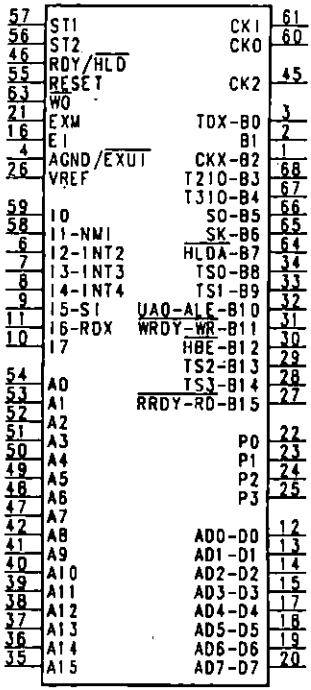


10480

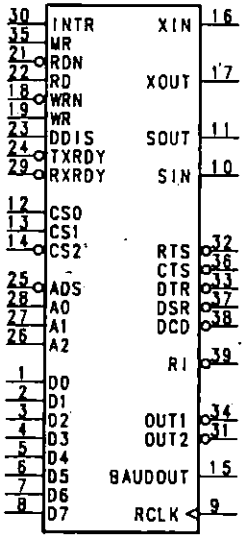
DRAWN BY: LD	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: DC	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 1XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 39 OF 113



1372

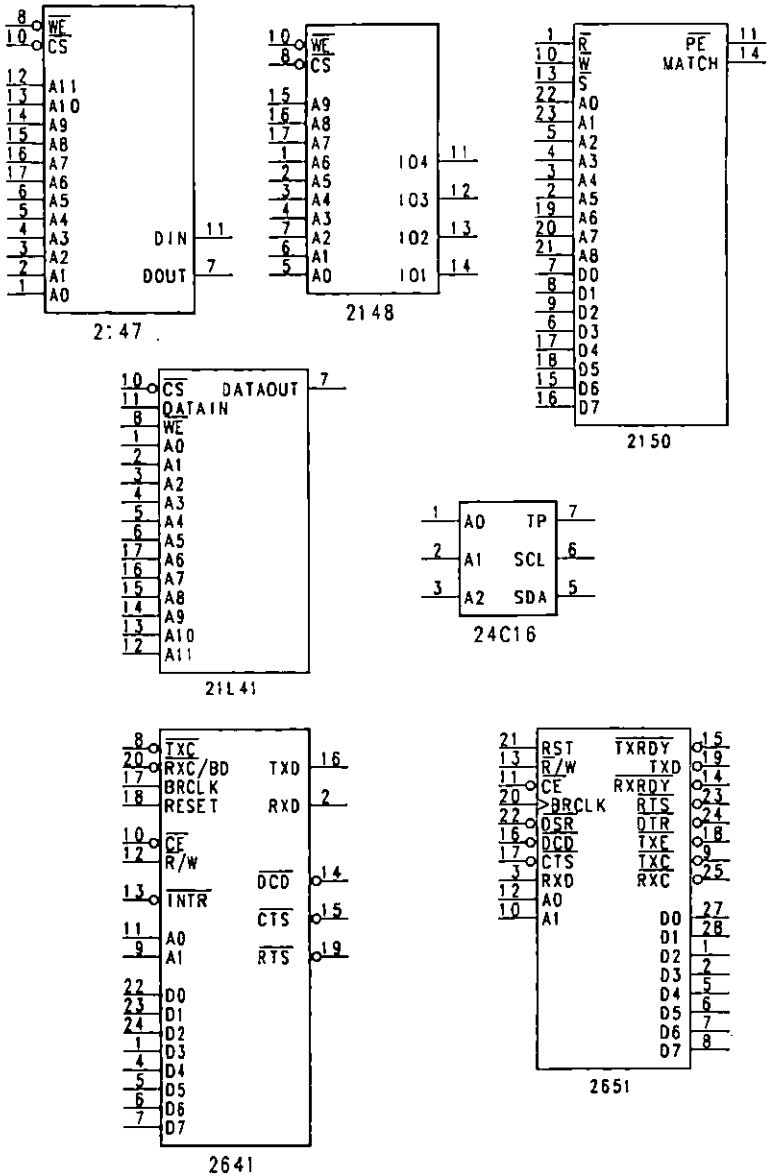


16164

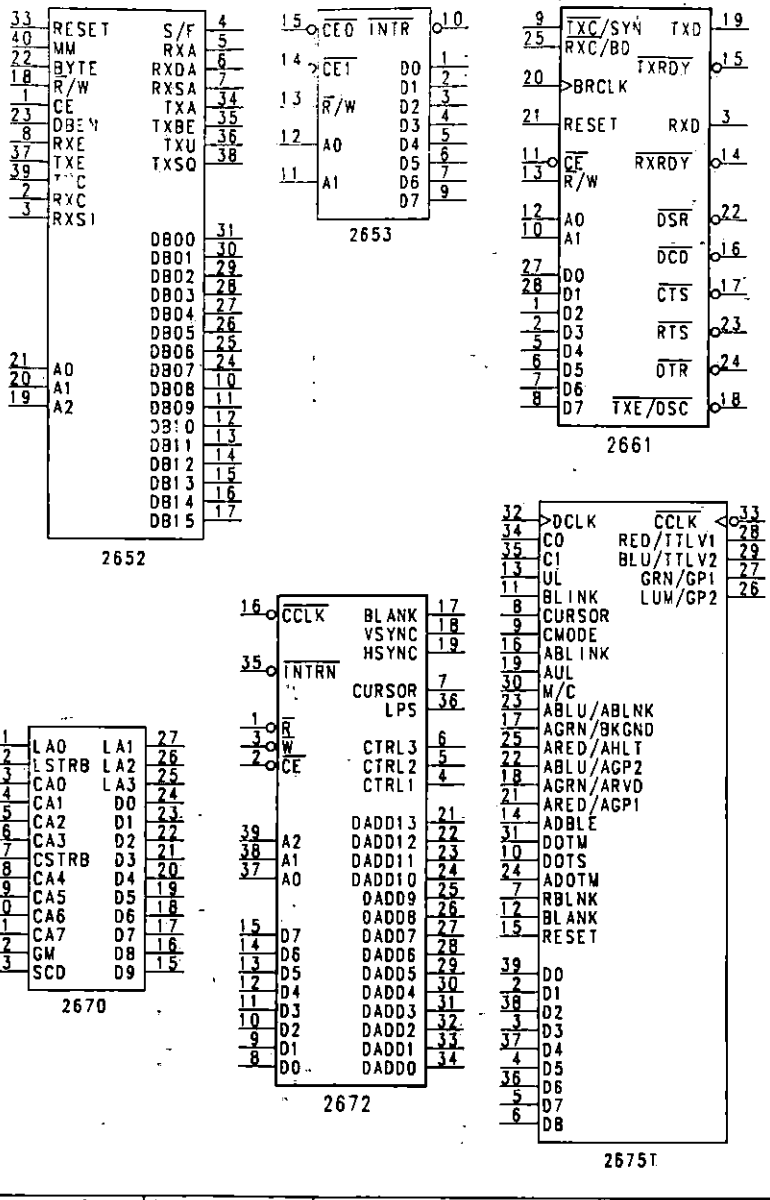


16550

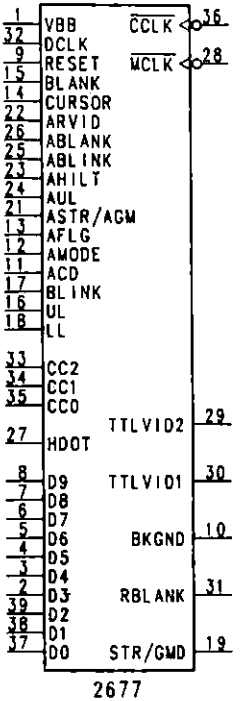
DRAWN BY: LP	DATE 1-3 90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: RP	DATE 1-3.90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 1XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 40 OF 113



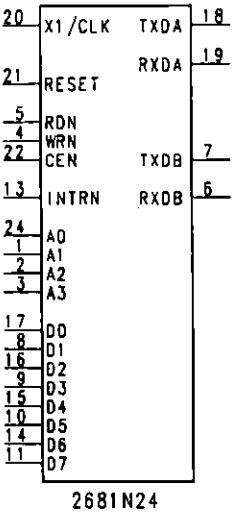
DRAWN BY: JP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 2XXXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 41 OF 113



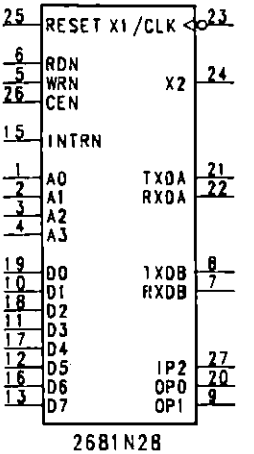
DRAWN BY: LP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: DC	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 2XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 42 OF 113



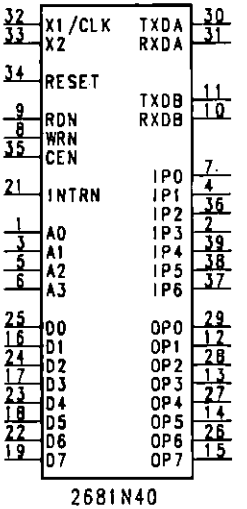
2677



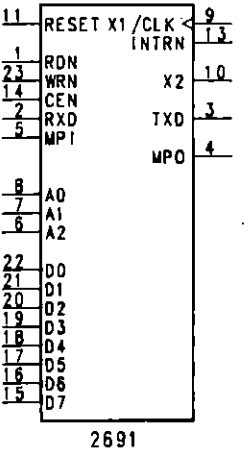
2681N24



2681N28

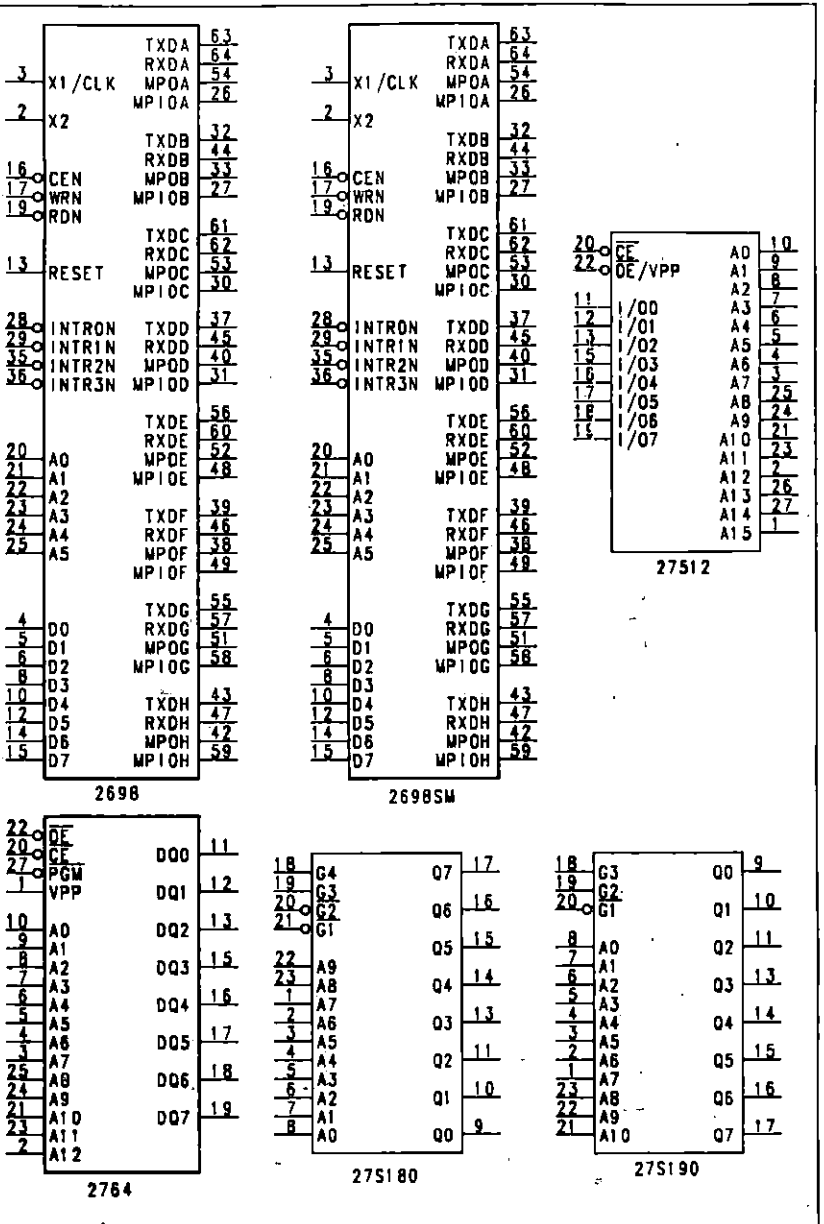


2681N40

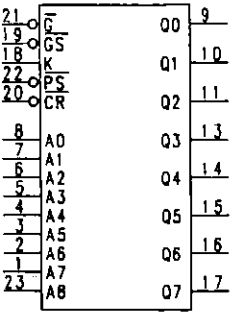


2691

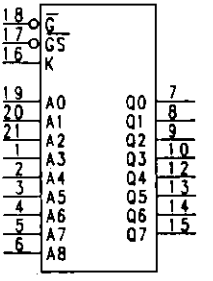
DRAWN BY: LP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: RK	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 2XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 43 OF 113



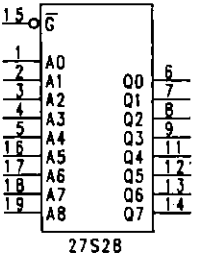
DRAWN BY: J.P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460	
CHECKED BY: TC	DATE 1-3-90		
APPROVED BY:	DATE 1-3-90	TITLE 2XXXXXX	
REVISION:	DATE		
JOB NAME:		REV. A	SCALE 1:1	SHEET 44 OF 113



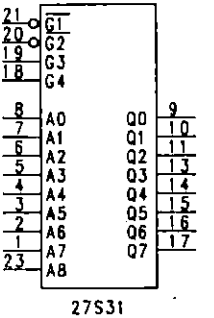
27S25



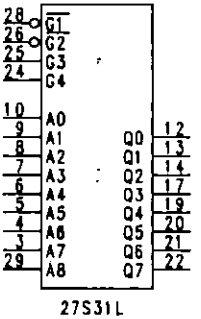
27S27



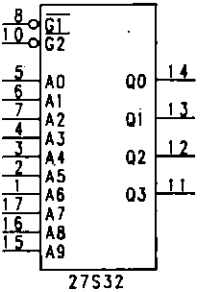
27S28



27S31

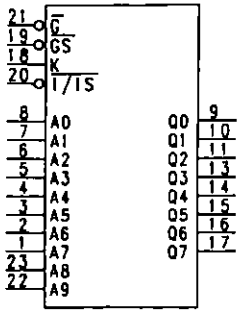


27S31L

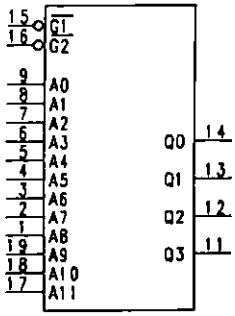


27S32

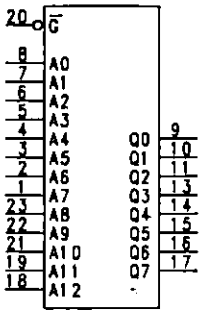
DRAWN BY: J.B.	DATE 1-3-90	CAD. SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 2XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 45 OF 113



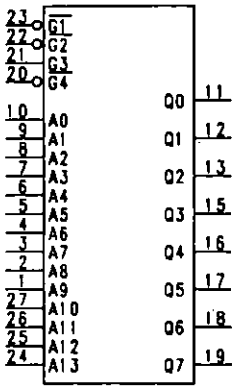
27S35



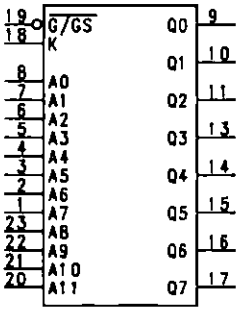
27S41



27S49

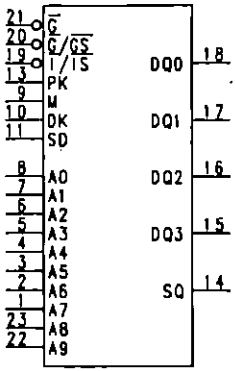


27S51

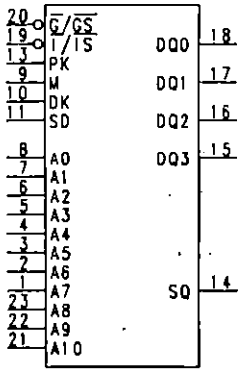


27S55

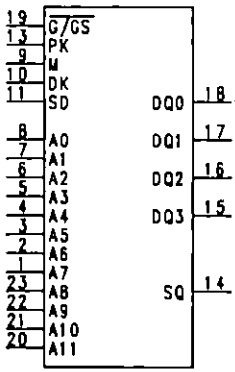
DRAWN BY: J.P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: (illegible)	DATE 1-3-90			
APPROVED BY: (illegible)	DATE 1-3-90	TITLE 2XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 46 OF 113



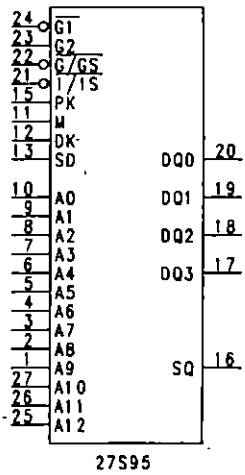
27S65



27S75

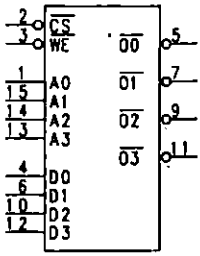


27S85

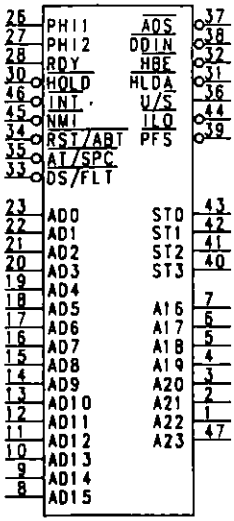


27S95

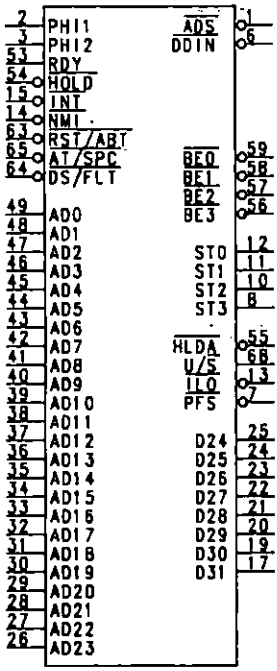
DRAWN BY: ZP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 2XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 47 OF 113



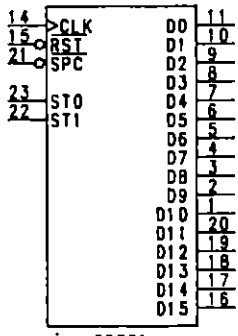
31L01



32016

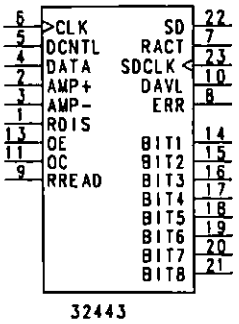


32032

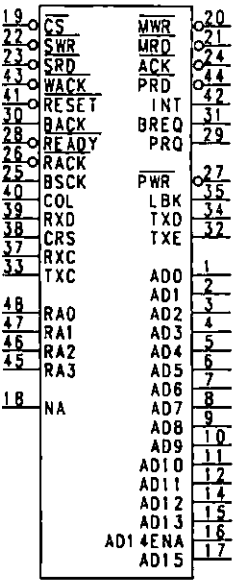


32081

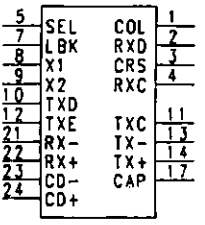
DRAWN BY: L.P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: DC	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 3XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 48 OF 113



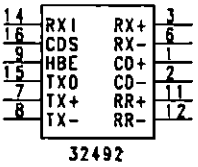
32443



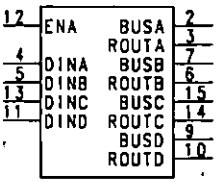
32490



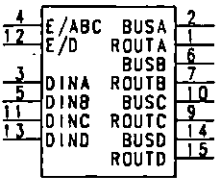
32491



32492

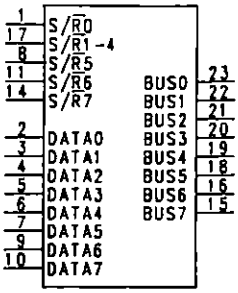


3440

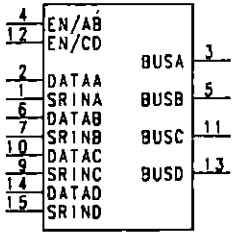


3446

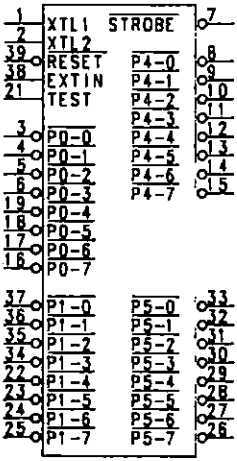
DRAWN BY: L P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 3XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 51 OF 113



3447

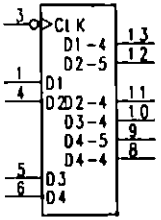


3448

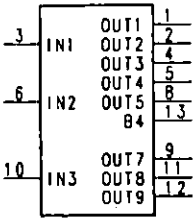


3870

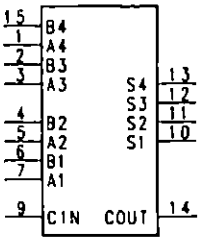
DRAWN BY: L.P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460			
CHECKED BY:	DATE 1-3-90				
APPROVED BY:	DATE 1-3-90	TITLE 3XXXXXX			
REVISION:	DATE				
JOB NAME:		REV. A	SCALE 1:1	SHEET 52 OF 113	



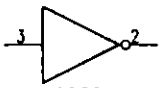
4006

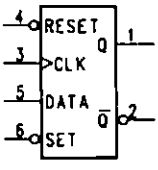


4007

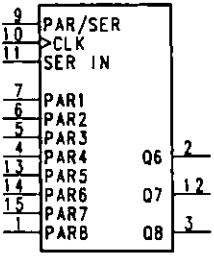


4008

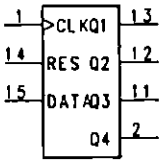

4009



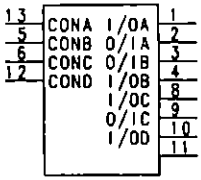
4013



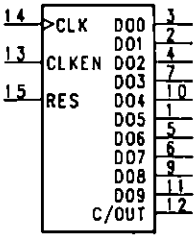
4014



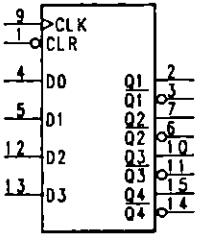
4015



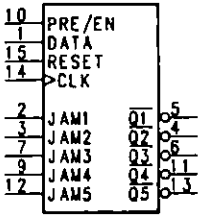
4016



4017

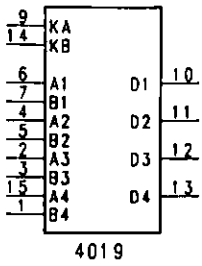


40175

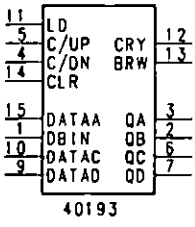


4018

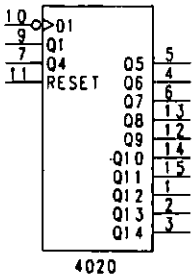
DRAWN BY: [illegible]	DATE 1390	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: [illegible]	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 4XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 53 OF 113



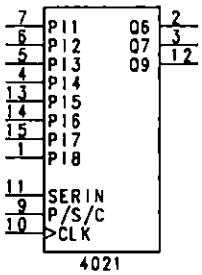
4019



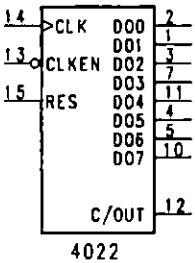
40193



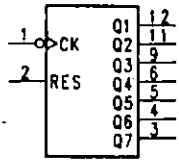
4020



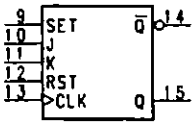
4021



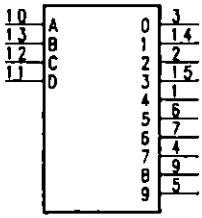
4022



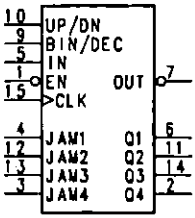
4024



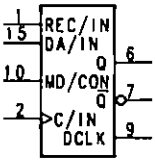
4027



4028

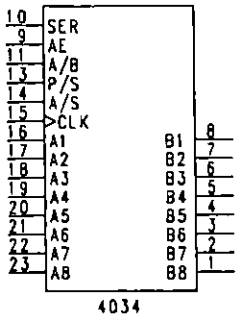


4029

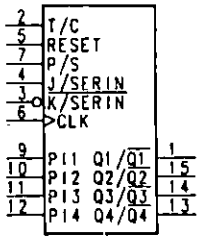


4031

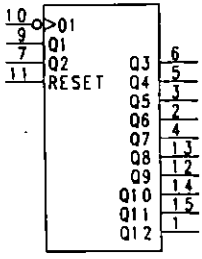
DRAWN BY: LP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: TC	DATE 1-3-90			
APPROVED BY: MH	DATE 1-3-90	TITLE 4XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 54 OF 113



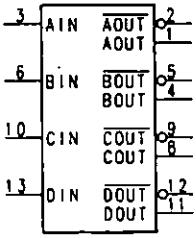
4034



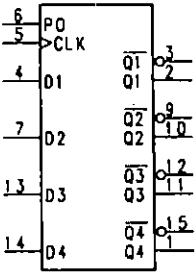
4035



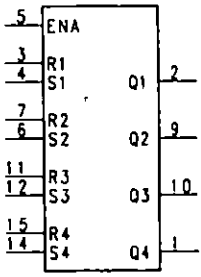
4040



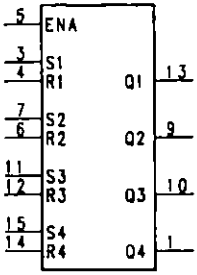
4041



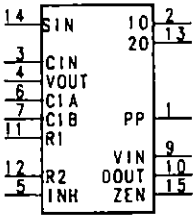
4042



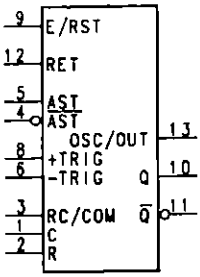
4043



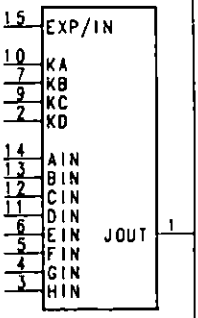
4044



4046

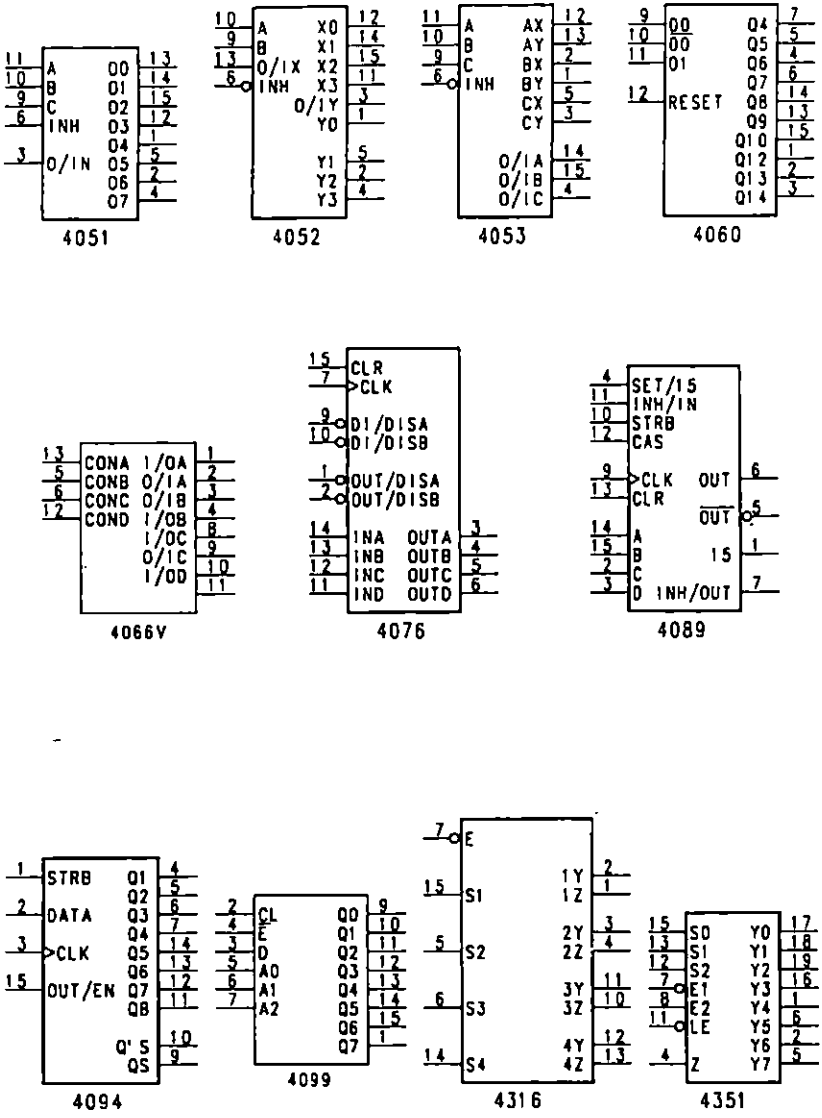


4047

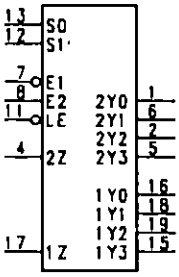


4048

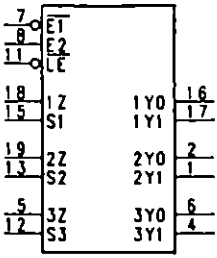
DRAWN BY: JP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-9-90	TITLE 4XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 55 OF 113



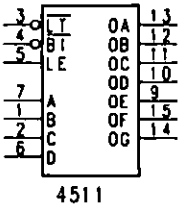
DRAWN BY: L. P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1.3.90			
APPROVED BY:	DATE 1-9-90	TITLE 4XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 56 OF 113



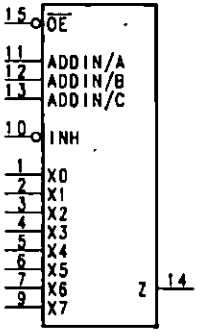
4352



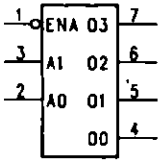
4353



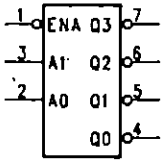
4511



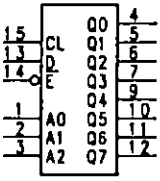
4512



4555

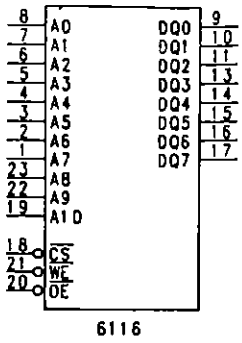


4556

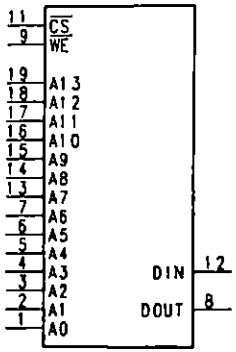


4724

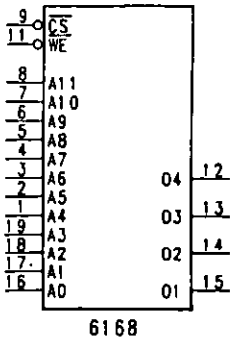
DRAWN BY: JP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1.3.90			
APPROVED BY:	DATE 1-3-90	TITLE 4XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 57 OF 113



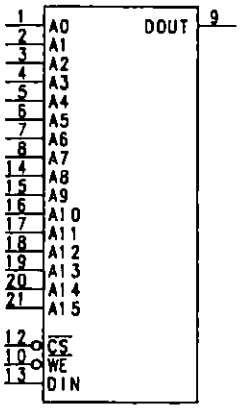
6116



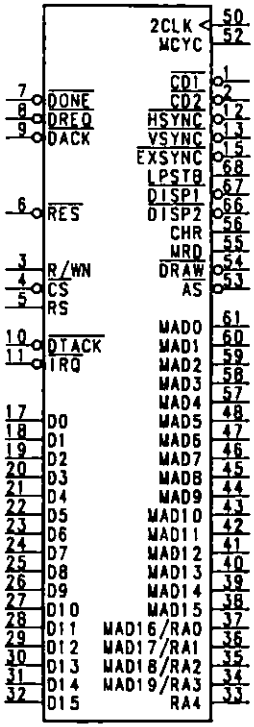
6167



6168

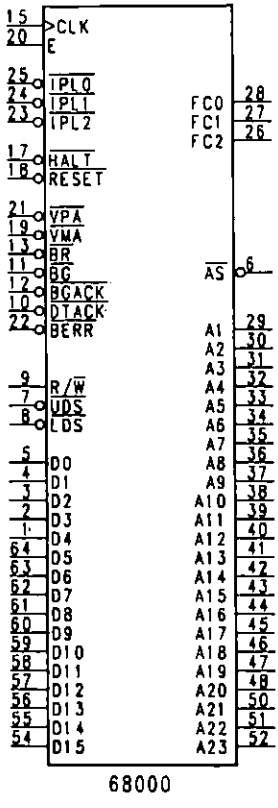


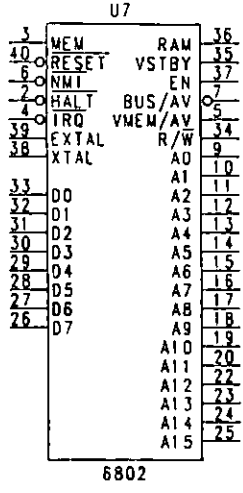
6287

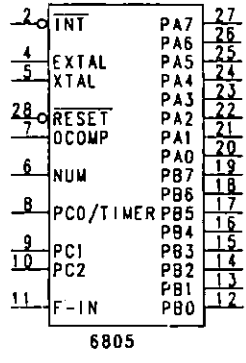


63484

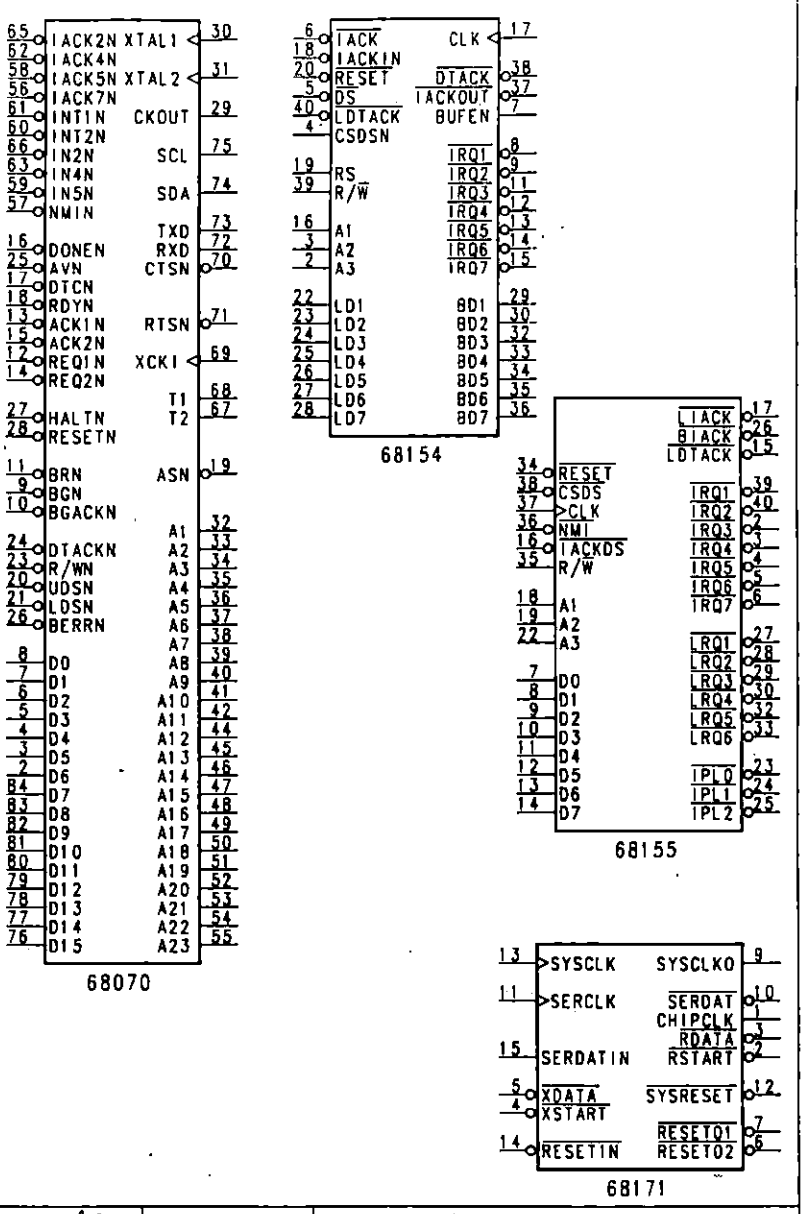
DRAWN BY: J.P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1.390			
APPROVED BY:	DATE 1-3-90	TITLE 6XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 58 OF 113



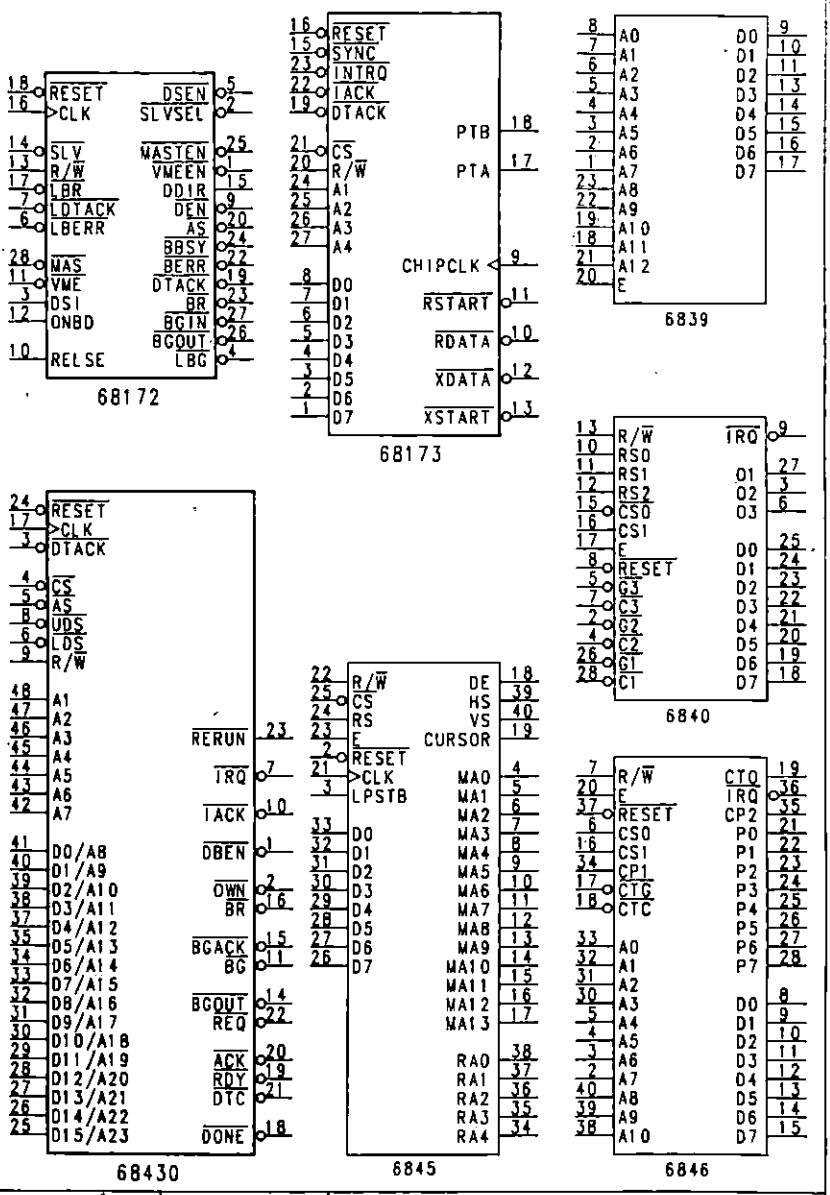




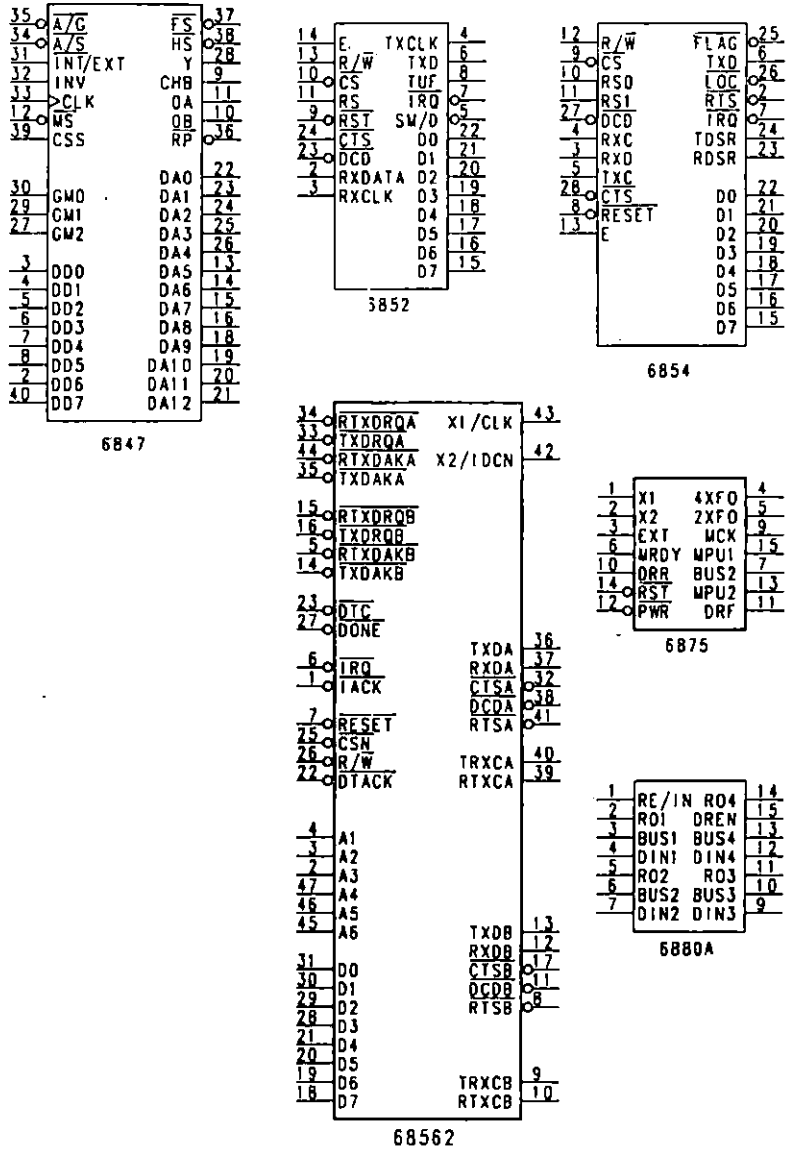
DRAWN BY: FP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: (DD)	DATE 1.3.90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 6XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 59 OF 113



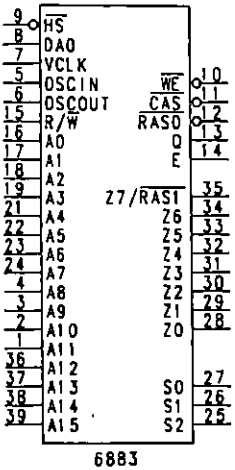
DRAWN BY: J.P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 6XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 60 OF 113



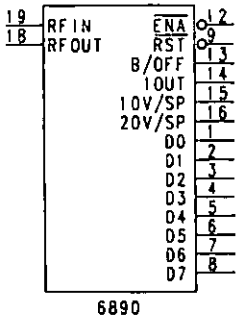
DRAWN BY: LP	DATE 1-3-90	CAD SOFTWARE INC.	
CHECKED BY:	DATE 1-3-90	119 RUSSELL STREET LITTLETON, MA 01460	
APPROVED BY:	DATE 1-3-90	TITLE 6XXXXXX	
REVISION:	DATE		
JOB NAME:	REV. A	SCALE 1:1	SHEET 61 OF 113



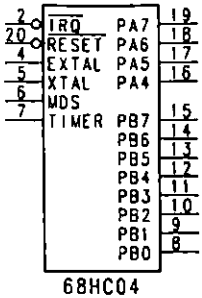
DRAWN BY: J P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460	
CHECKED BY:	DATE 1-3-90		
APPROVED BY:	DATE 1-3-90	TITLE 6XXXXXX	
REVISION:	DATE		
JOB NAME:		REV. A	SCALE 1:1	SHEET 62 OF 113



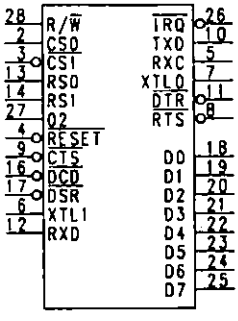
6883



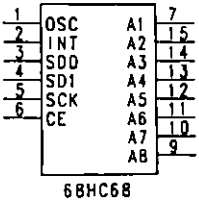
6890



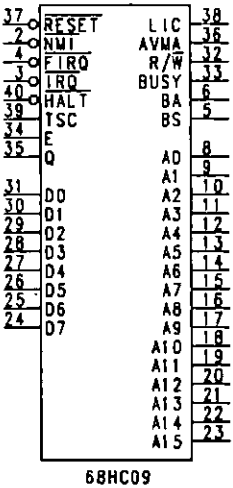
68HC04



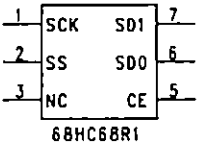
68HC51



68HC68

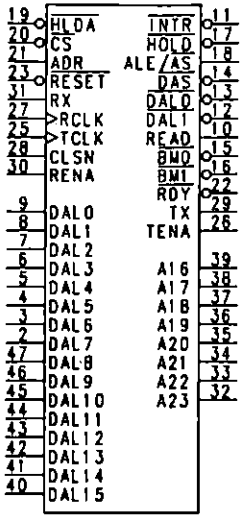


68HC09

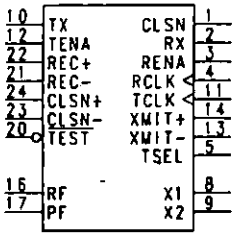


68HC68R1

DRAWN BY: J P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: CC	DATE 1.3.90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 6XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 63 OF 113

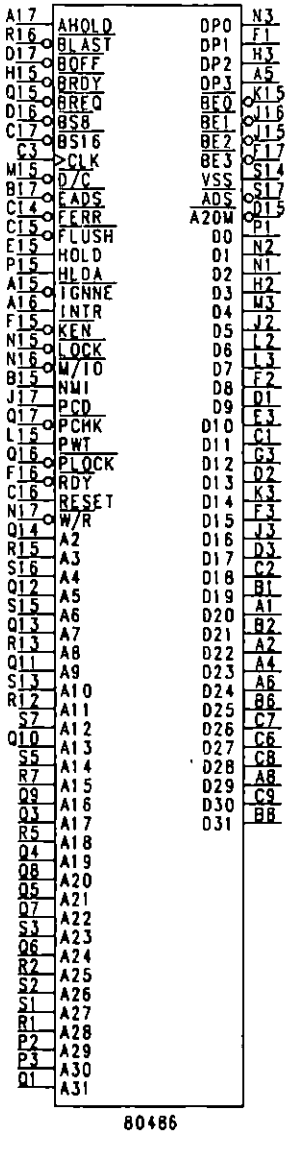


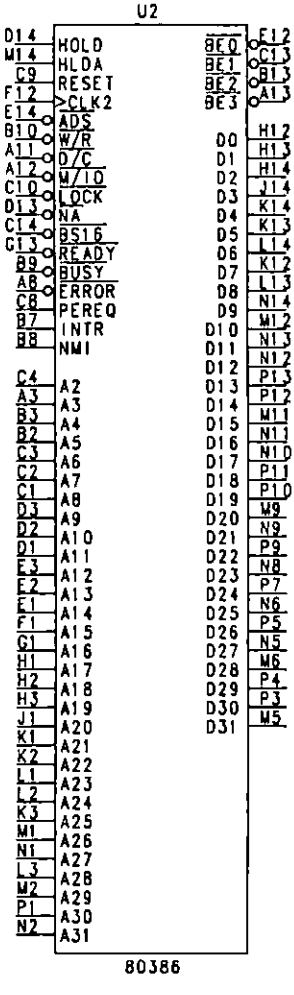
7990



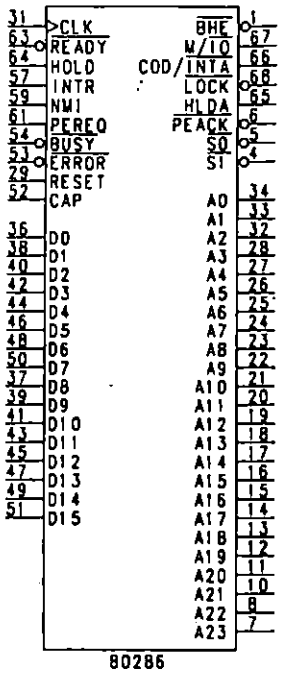
7992

DRAWN BY: LP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: DC	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 7XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 64 OF 113

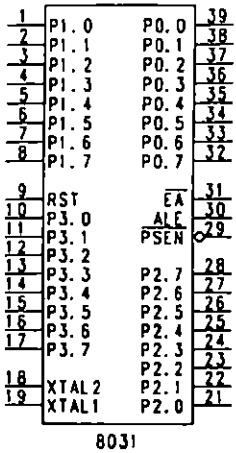




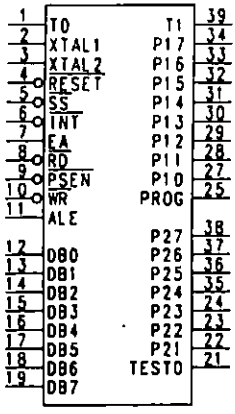
DRAWN BY: J P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1.3.90			
APPROVED BY:	DATE 1-3-90	TITLE 8XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 65 OF 113



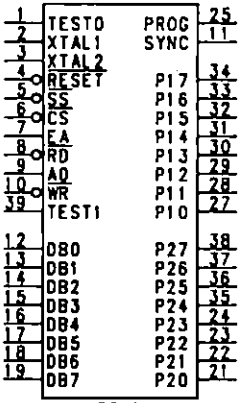
80286



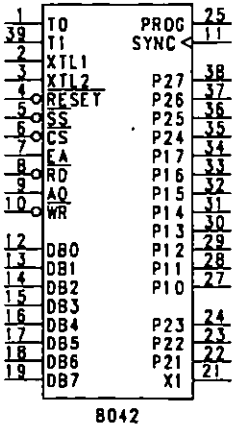
8031



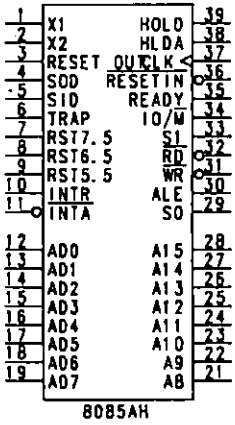
8035AHL



8041

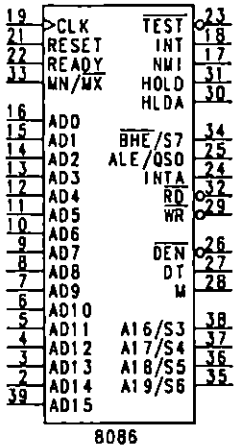


8042

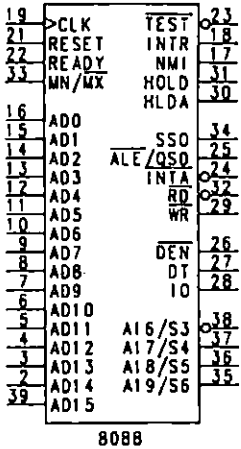


8085AH

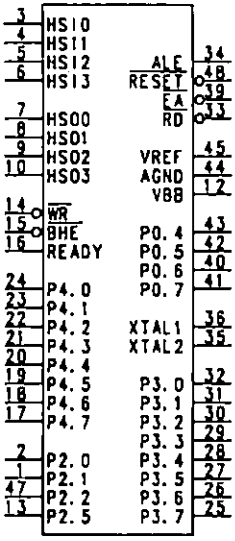
DRAWN BY: LP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 8XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 66 OF 113



8086

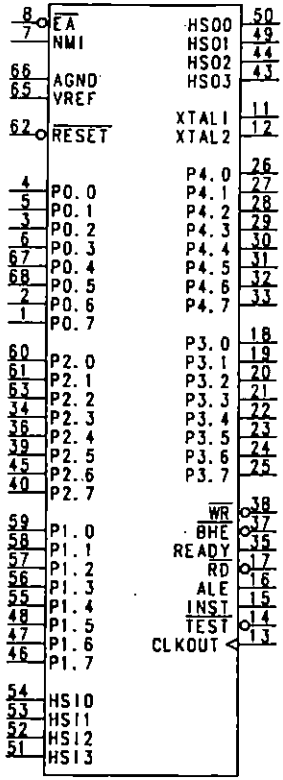


8088

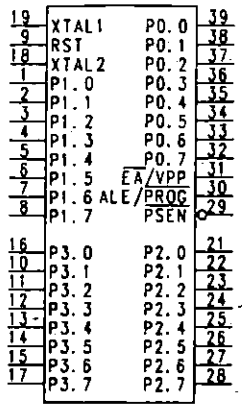


8094

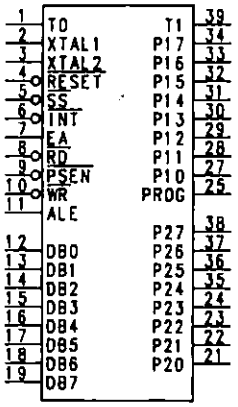
DRAWN BY: JP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: TC	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 8XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 67 OF 113



8096

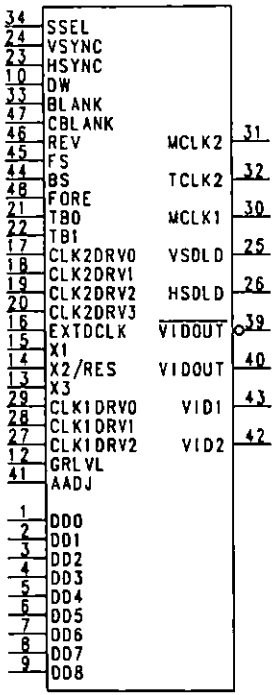


80C252

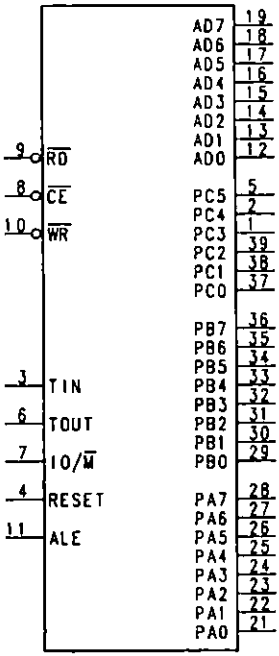


80C39

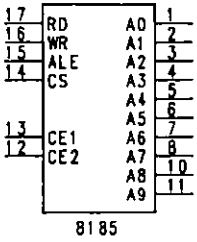
DRAWN BY: LP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: [illegible]	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 8XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 68 OF 113



8152

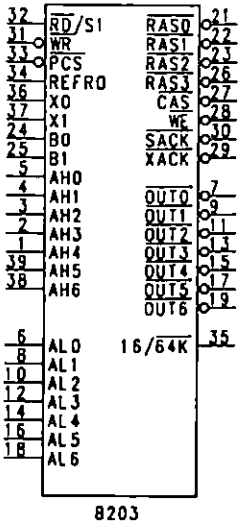


8155H

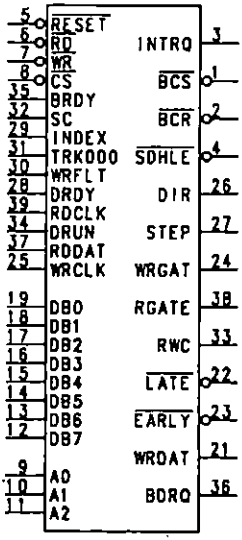


8185

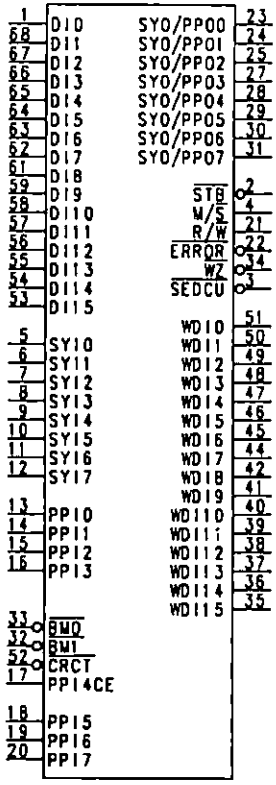
DRAWN BY: J.P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: DC	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 8XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 69 OF 113



8203

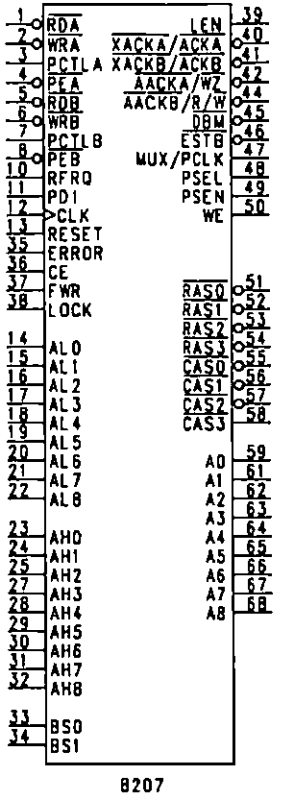


82064

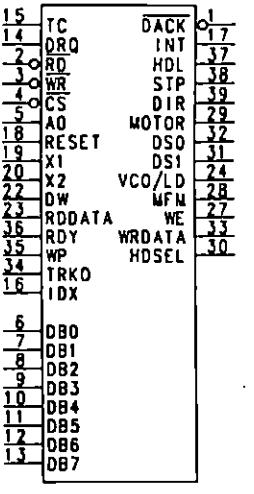


8206

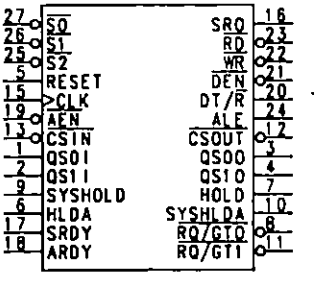
DRAWN BY: K. P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: [illegible]	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 8XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 70 OF 113



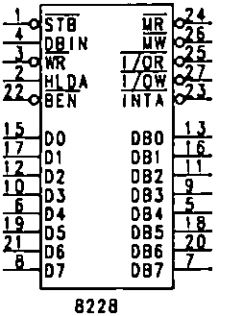
8207



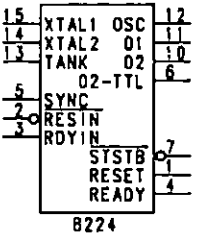
82072



82188

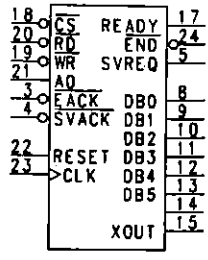


8228

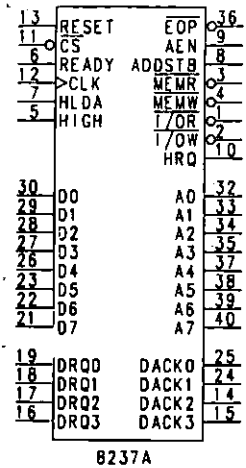


8224

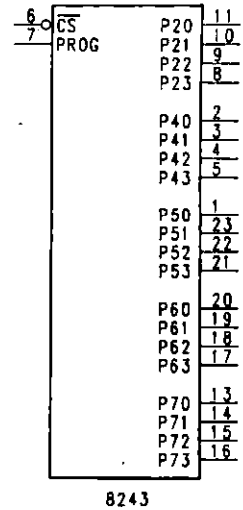
DRAWN BY: L.P.	DATE 1.3.90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: [illegible]	DATE 1.3.90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 8XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 71 OF 113



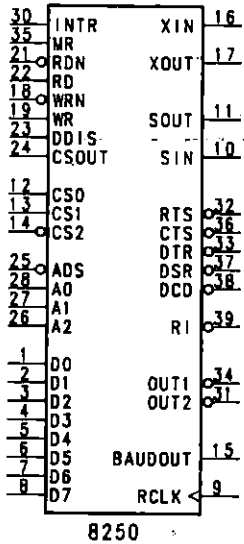
8231



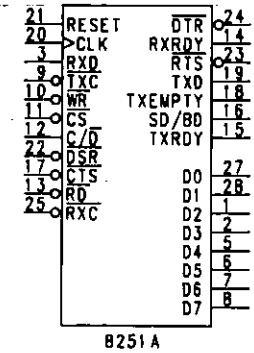
8237A



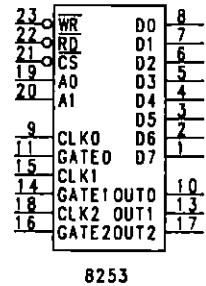
8243



8250

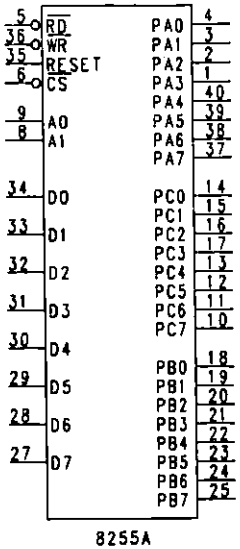


8251A

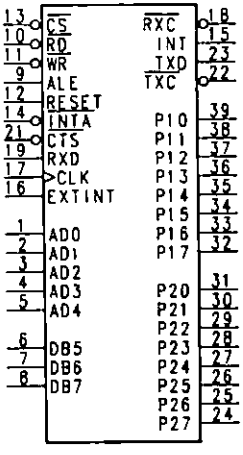


8253

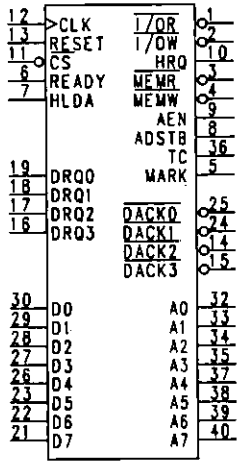
DRAWN BY: L.P.	DATE 1.3.90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 8XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 72 OF 113



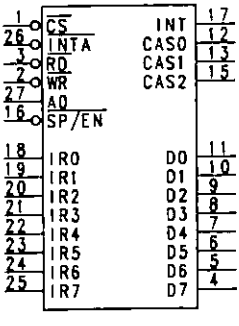
8255A



8256AH

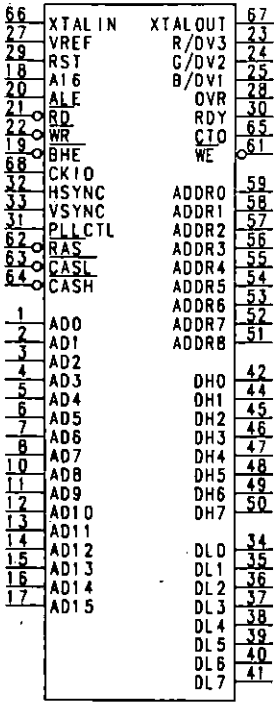


8257

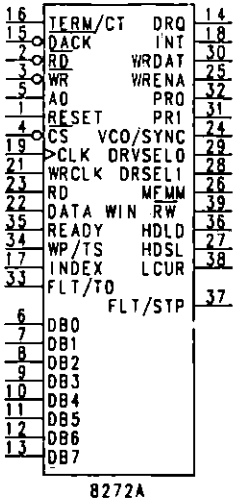


8259A

DRAWN BY: L P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: (D)	DATE 1.3.90			
APPROVED BY: Mgd	DATE 1-3-90	TITLE 8XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 73 OF 113

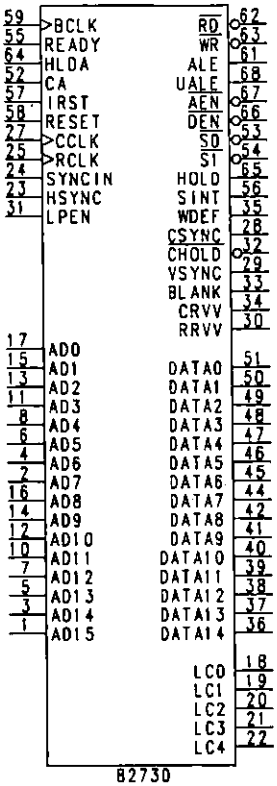


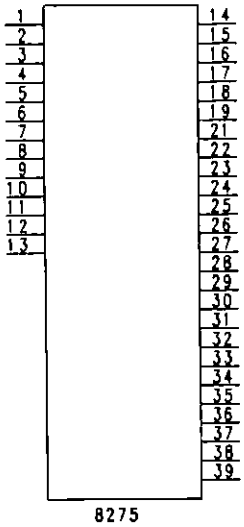
82716

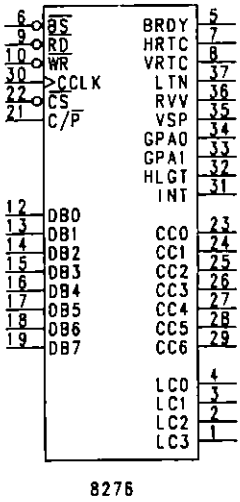


8272A

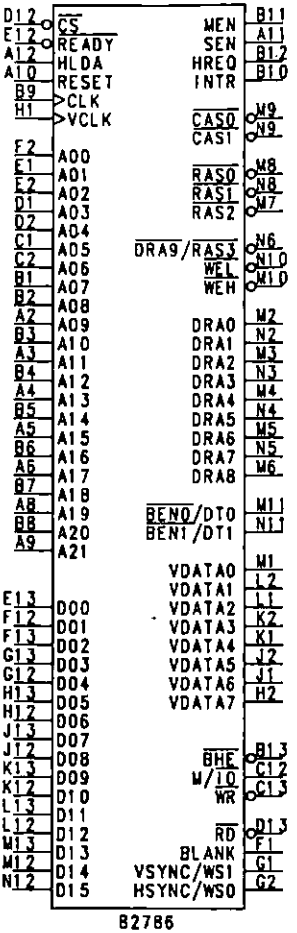
DRAWN BY: LP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 8XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 74 OF 113



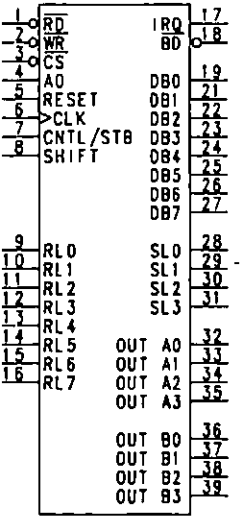




DRAWN BY: JP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1.3.90			
APPROVED BY:	DATE 1-3-90	TITLE 8XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 75 OF 113

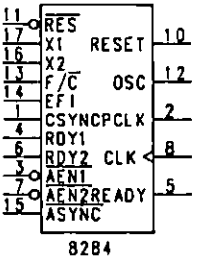


82786

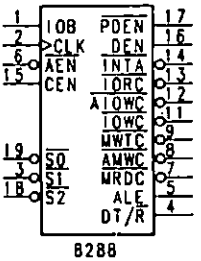


8279

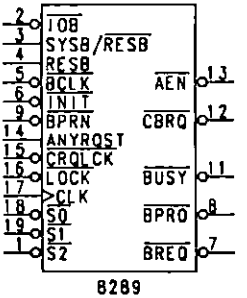
DRAWN BY: L.P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460			
CHECKED BY:	DATE 1.3.90				
APPROVED BY:	DATE 1-3-90	TITLE 8XXXXXX			
REVISION:	DATE				
JOB NAME:		REV. A	SCALE 1:1	SHEET 76 OF 113	



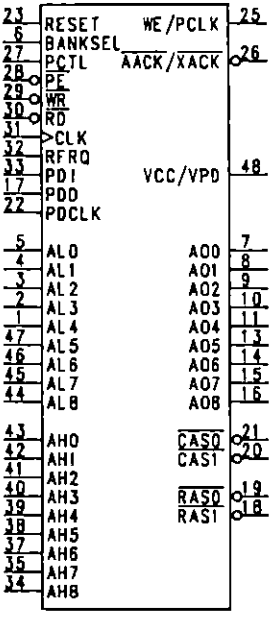
8284



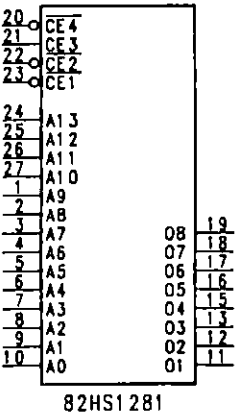
8288



8289

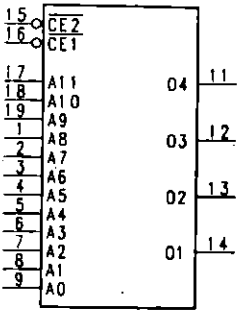


82C08

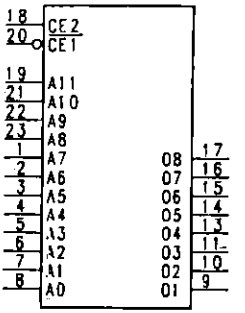


82HS1281

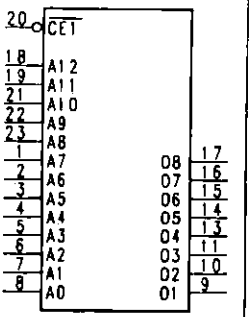
DRAWN BY: LP.	DATE 1.3.90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 8XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 77 OF 113



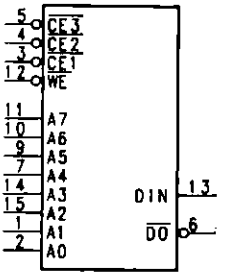
82HS195



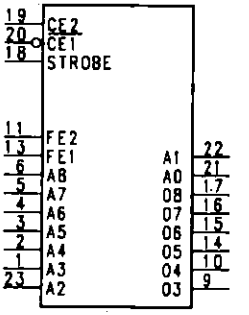
82HS321



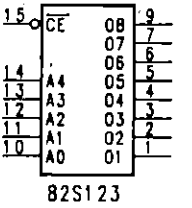
82HS641



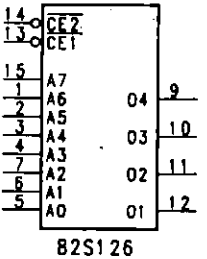
82LS16



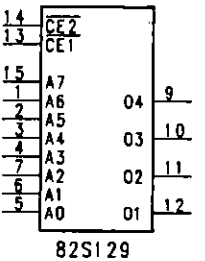
82S115



82S123

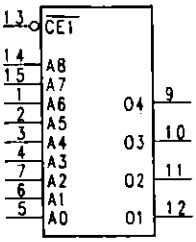


82S126

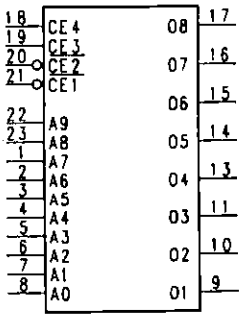


82S129

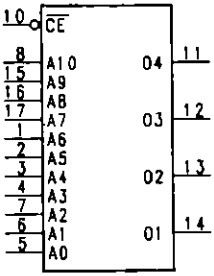
DRAWN BY: L.P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 8XXXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 78 OF 113



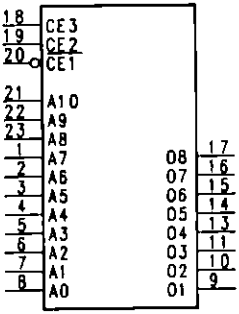
82S130



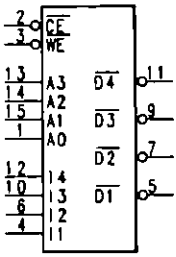
82S181



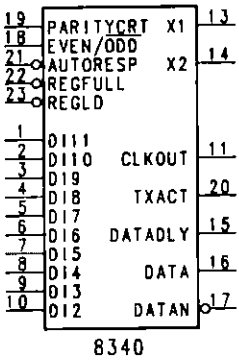
82S185



82S191

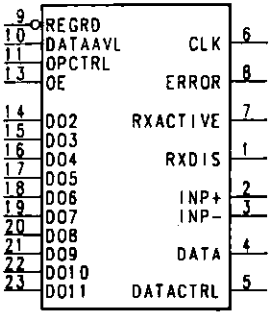


82S25

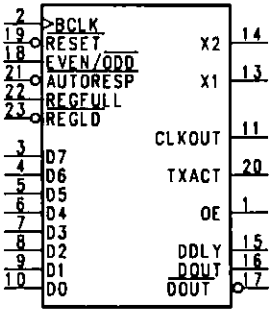


8340

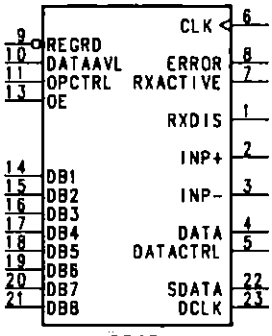
DRAWN BY: JP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 8XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 79 OF 113



8341

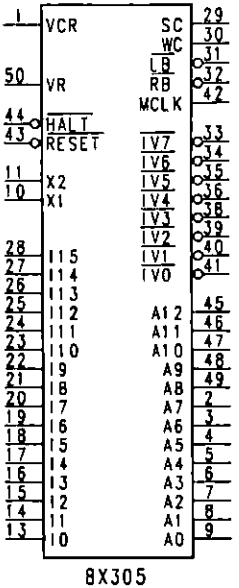


8342

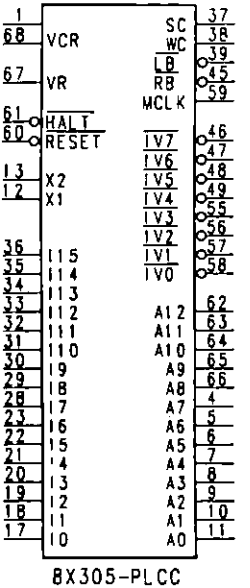


8343

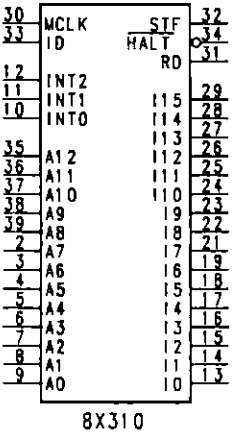
DRAWN BY: LP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 8XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 80 OF 113



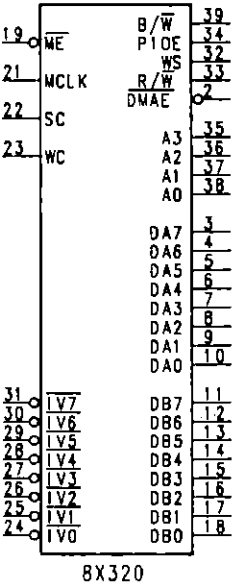
8X305



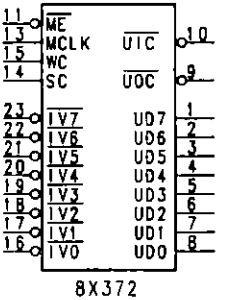
8X305-PLCC



8X310

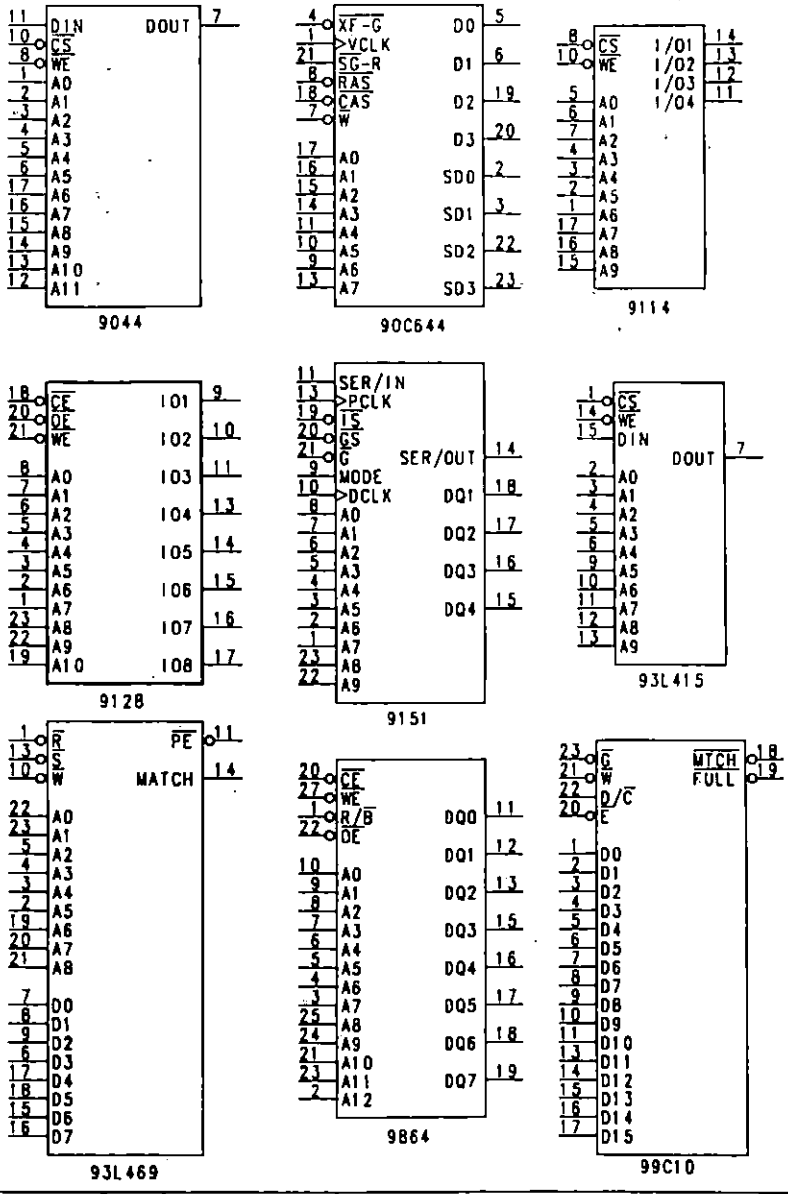


8X320

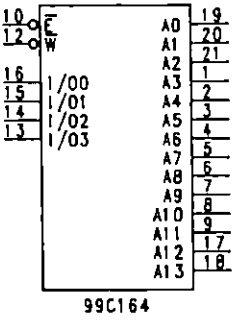


8X372

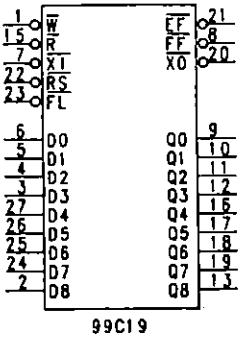
DRAWN BY: JP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: RD	DATE 1.3.90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 8XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 81 OF 113



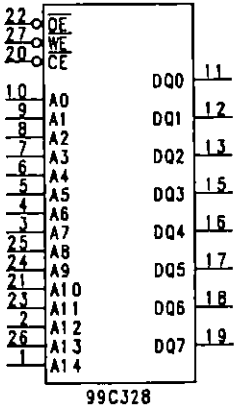
DRAWN BY: J P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460.		
CHECKED BY:	DATE 1.3.90			
APPROVED BY:	DATE 1-3-90	TITLE 9XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 82 OF 113



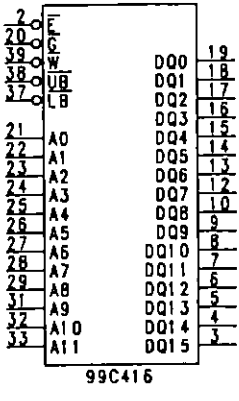
99C164



99C19

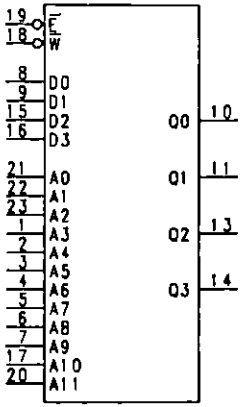


99C328

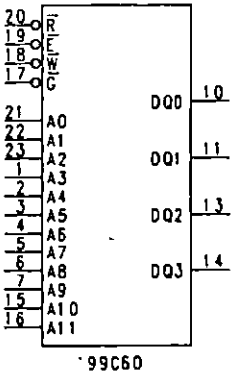


99C416

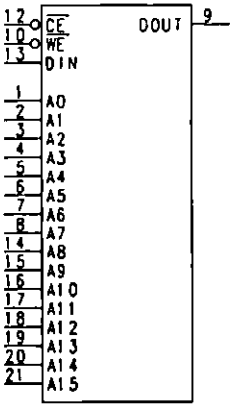
DRAWN BY: J P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1.3.90			
APPROVED BY:	DATE 1-3-90	TITLE 9XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 83 OF 113



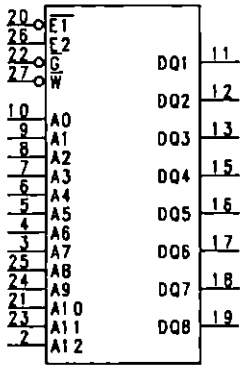
99C58



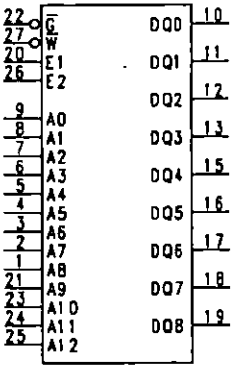
99C60



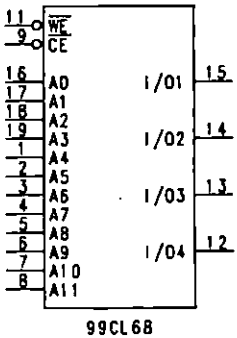
99C641



99C88

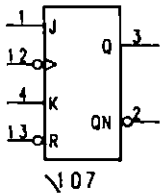


99C89

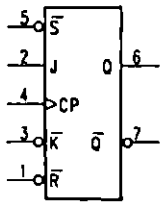


99CL68

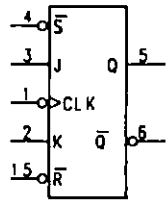
DRAWN BY: JP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 9XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 84 OF 113



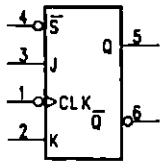
\107



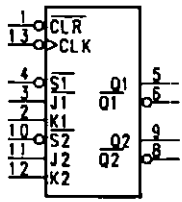
\109



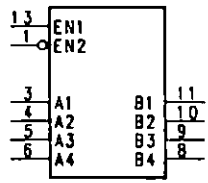
\112



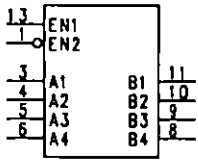
\113



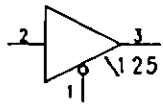
\114



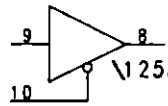
\1242



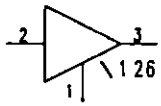
\1243



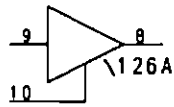
\125



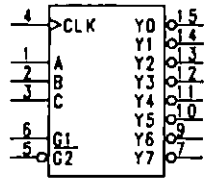
\125A



\126

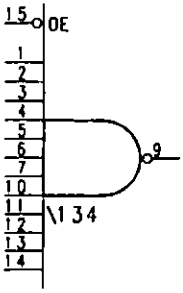


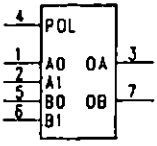
\126A

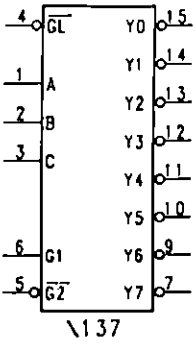


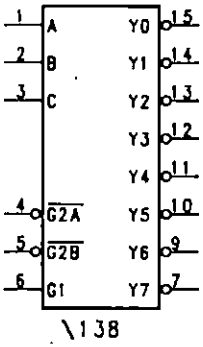
\131

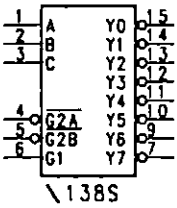
DRAWN BY: JP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: DC	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 54/74 SERIES (\XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 85 OF 113

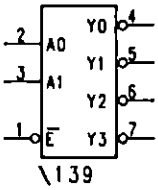


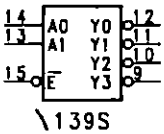


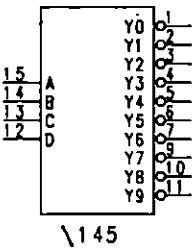




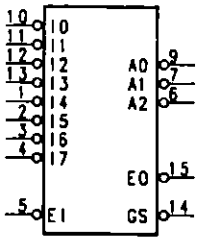




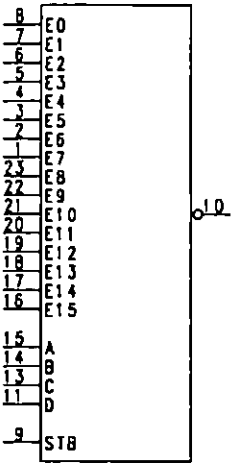




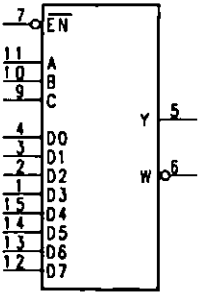
DRAWN BY: ZP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460			
CHECKED BY: DD	DATE 1-3-90				
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 54/74 SERIES (\XXX)			
REVISION:	DATE				
JOB NAME:		REV. A	SCALE 1:1	SHEET 86 OF 113	



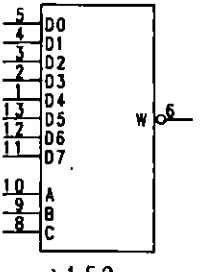
\148



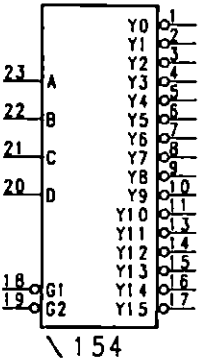
\150



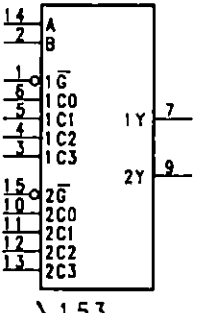
\151



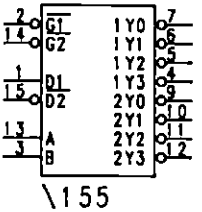
\152



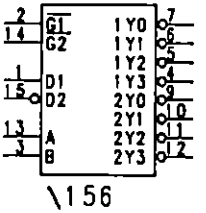
\154



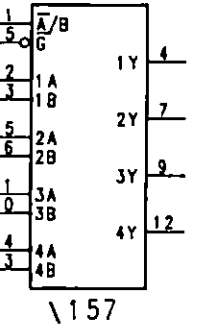
\153



\155

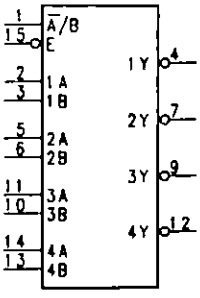


\156

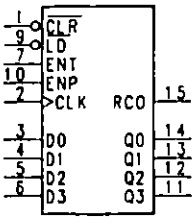


\157

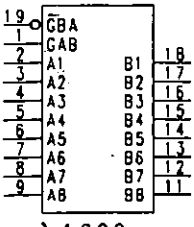
DRAWN BY: LP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: TM	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 54/74 SERIES (\XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 87 OF 113



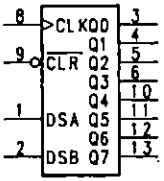
\158



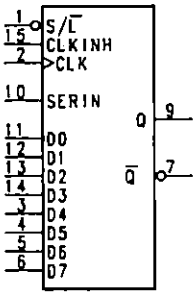
\160



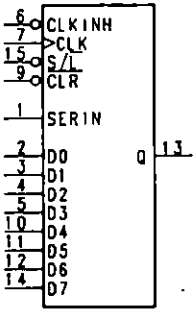
\1620



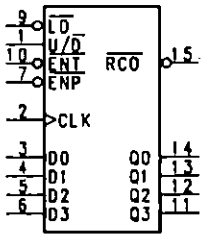
\164



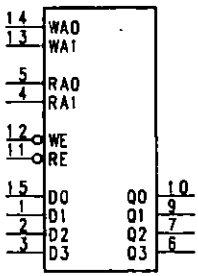
\165



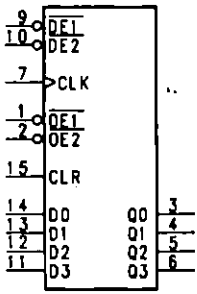
\166



\168

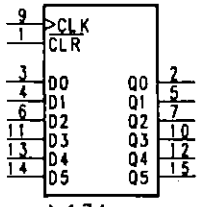


\170

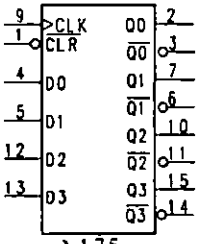


\173

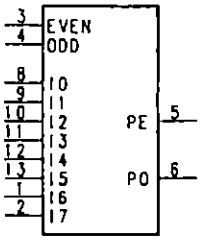
DRAWN BY: JP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: DC	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 54/74 SERIES (\XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 88 OF 113



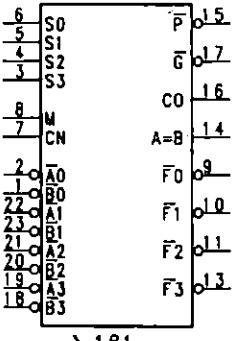
\174



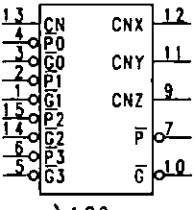
\175



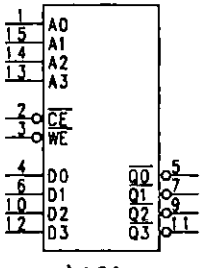
74180



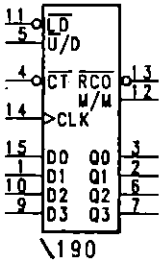
\181



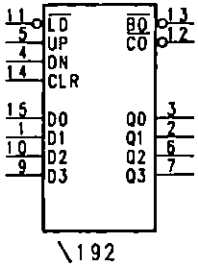
\182



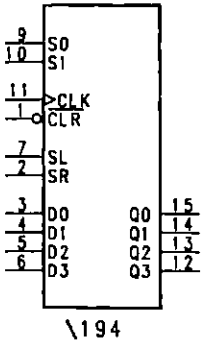
\189



\190

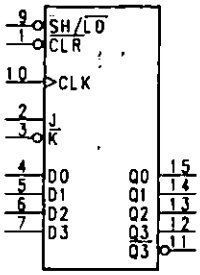


\192

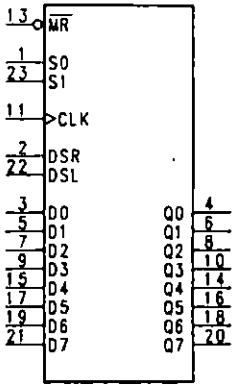


\194

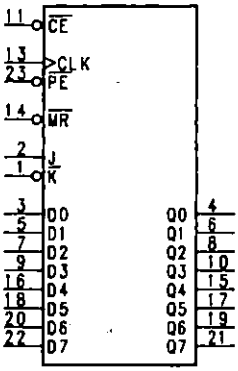
DRAWN BY: L P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 54/74 SERIES (\XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 89 OF 113



\195

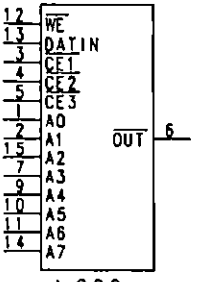


\198

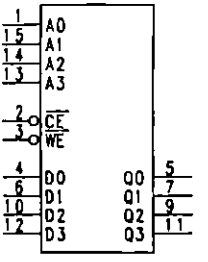


\199

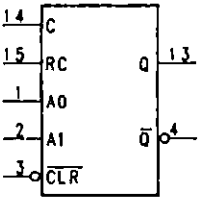
DRAWN BY: J P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460	
CHECKED BY:	DATE 1-3-90		
APPROVED BY:	DATE 1-3-90	TITLE 54/74 SERIES (\XXX)	
REVISION:	DATE		
JOB NAME:		REV. A	SCALE 1:1	SHEET 90 OF 113



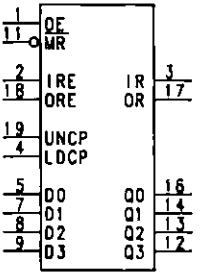
\200



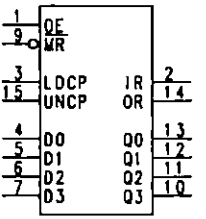
\219A



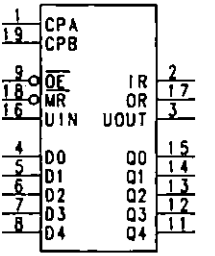
\221



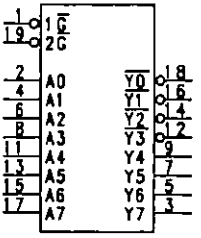
\222



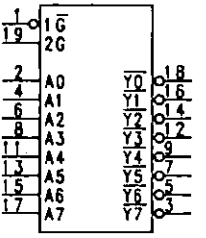
\224



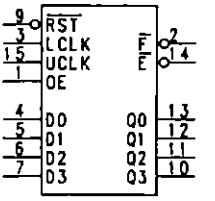
\225



\230

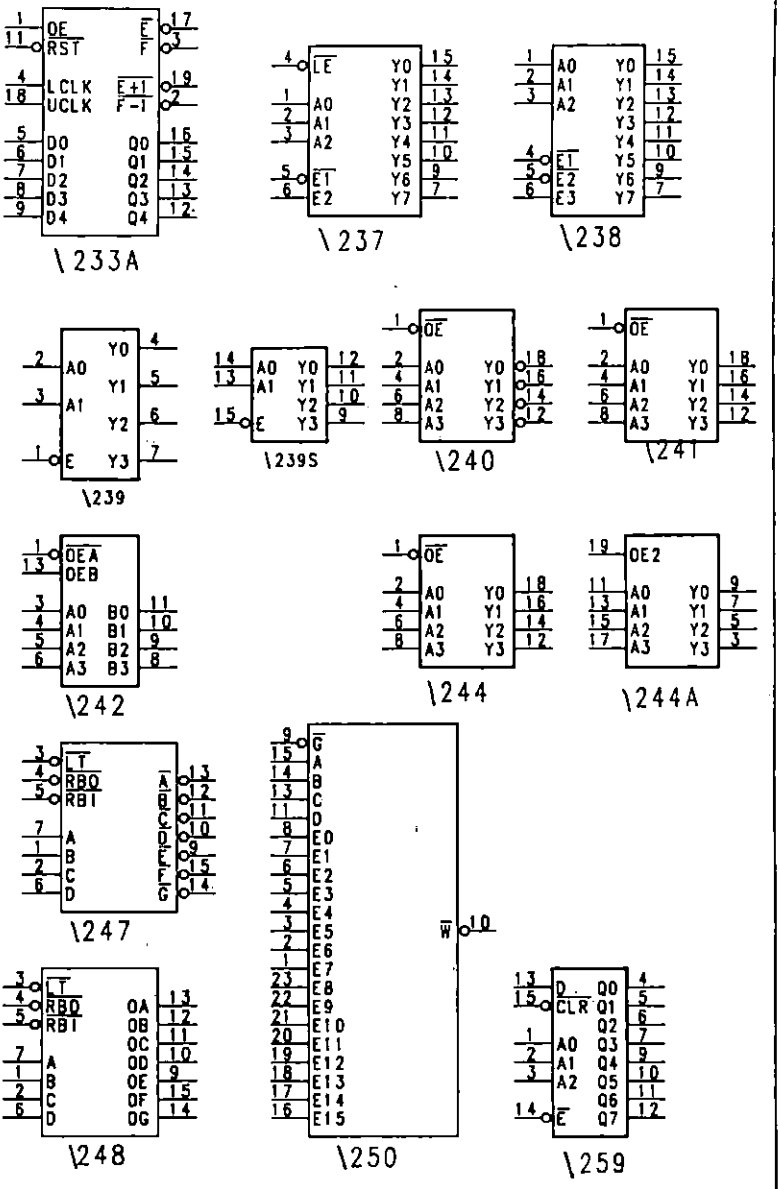


\231

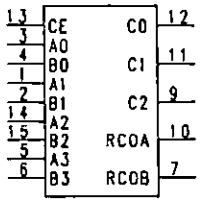


\232

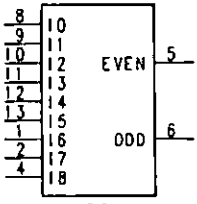
DRAWN BY: J.P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: [illegible]	DATE 1-3-90			
APPROVED BY:	DATE	TITLE 54/74 SERIES (\XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 91 OF 113



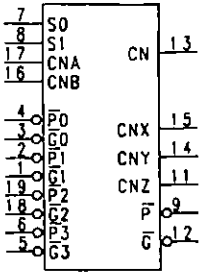
DRAWN BY: [illegible]	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: [illegible]	DATE 1-3-90			
APPROVED BY:	DATE	TITLE 54/74 SERIES (\XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 92 OF 113



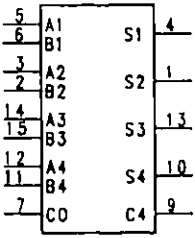
\264



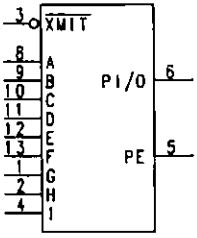
\280



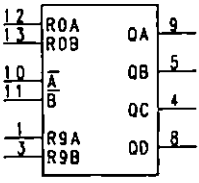
\282



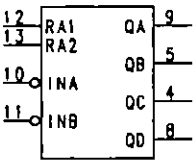
\283



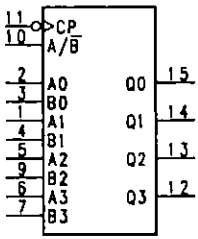
\286



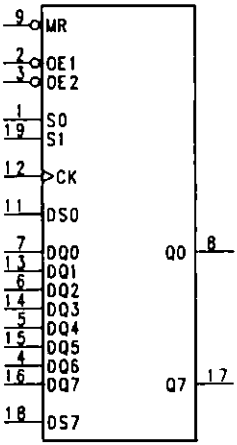
\290



\293

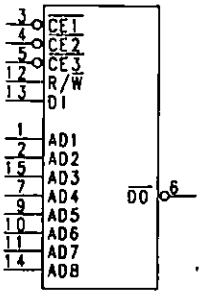


\298

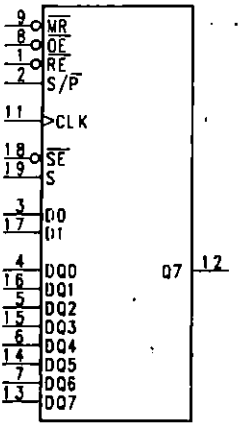


\299

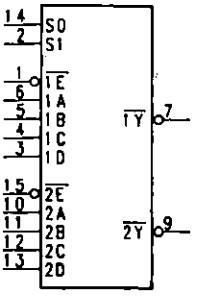
DRAWN BY: LP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 54/74 SERIES (\XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 93 OF 113



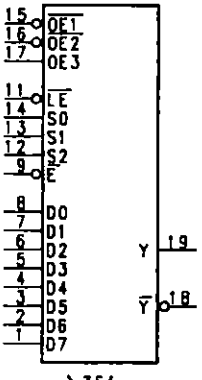
\301



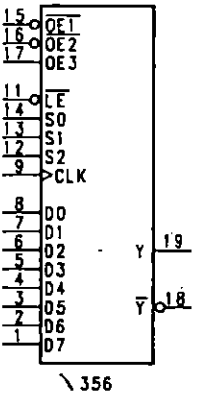
\322



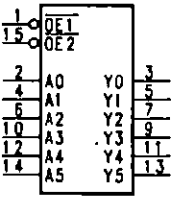
\352



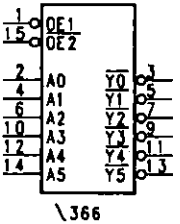
\354



\356

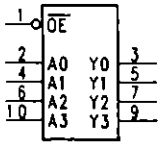


\365

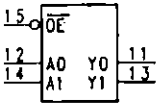


\366

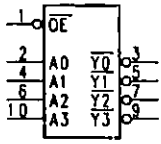
DRAWN BY: J P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: @	DATE 1-3-90			
APPROVED BY:	DATE	TITLE 54/74 SERIES (XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 94 OF 113



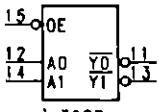
\ 367A



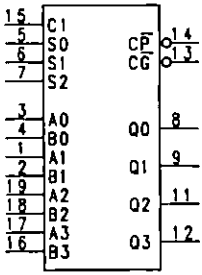
\ 367B



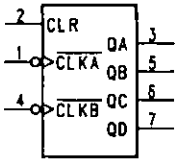
\ 368A



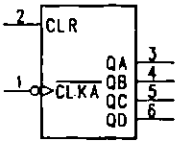
\ 368B



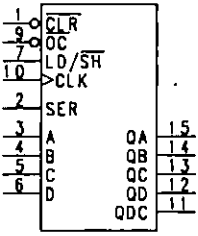
\ 381



\ 390

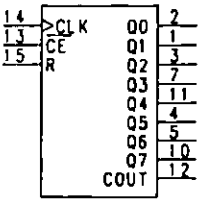


\ 393

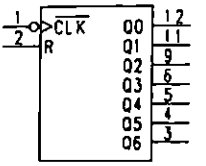


\ 395

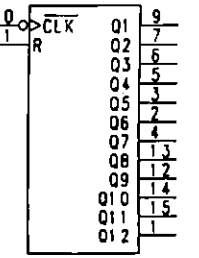
DRAWN BY: fp	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE	TITLE 54/74 SERIES (XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 95 OF 113



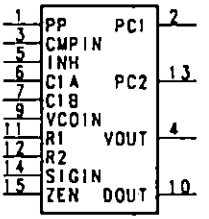
\4022



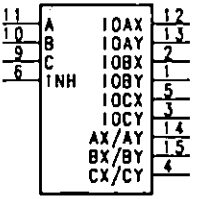
\4024



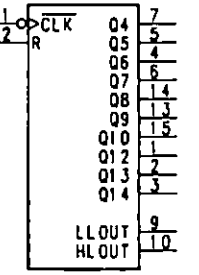
\4040



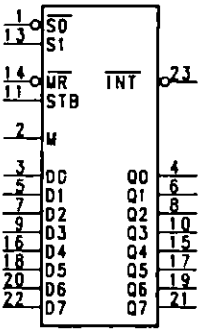
\4046



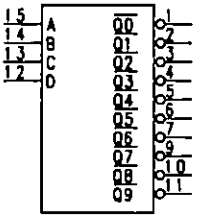
\4053



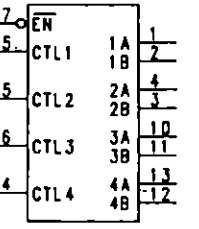
\4060



\412

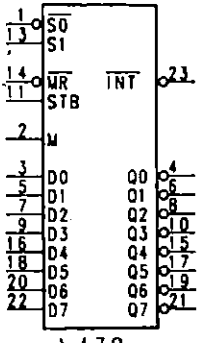


\42

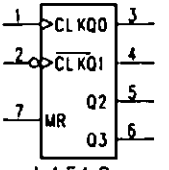


\4316

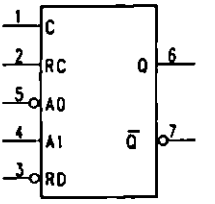
DRAWN BY: J.P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: @	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 54/74 SERIES (\XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 96 OF 113



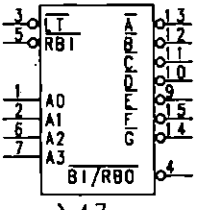
\432



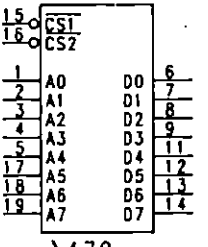
\4518



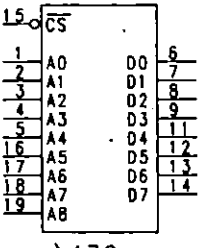
\4538



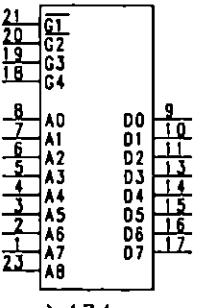
\47



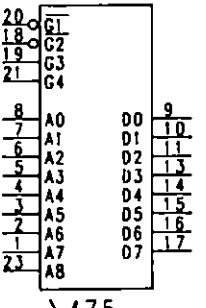
\470



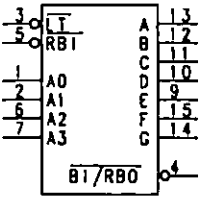
\472



\474

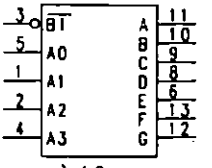


\475

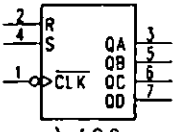


\48

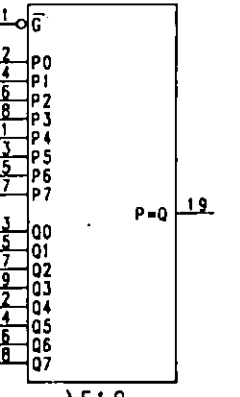
DRAWN BY: JP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460	
CHECKED BY: DC	DATE 1-3-90		
APPROVED BY: [illegible]	DATE 1-3-90	TITLE: 54/74 SERIES (\XXX)	
REVISION:	DATE		
JOB NAME:		REV. A	SCALE 1:1	SHEET 97 OF 113



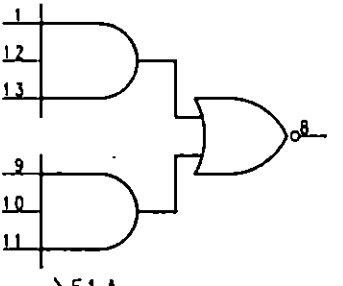
\49



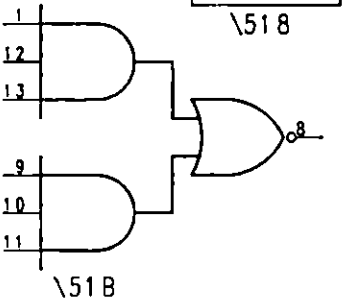
\490



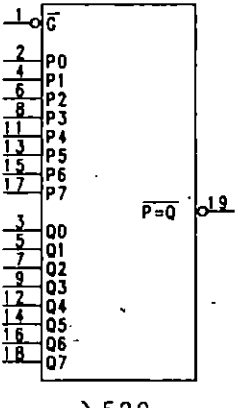
\518



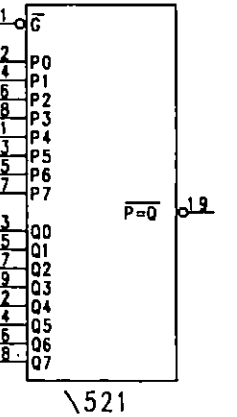
\51A



\51B

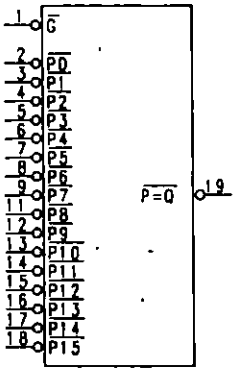


\520

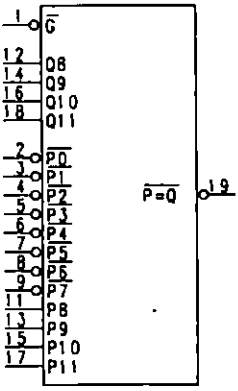


\521

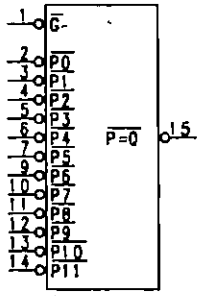
DRAWN BY: J.P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 54/74 SERIES (XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 98 OF 113



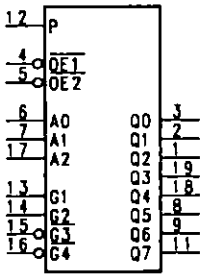
\526



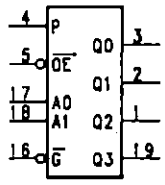
\527



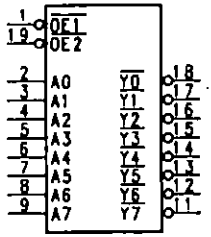
\528



\538

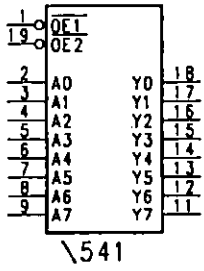


\539

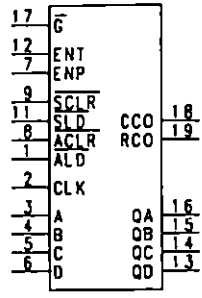


\540

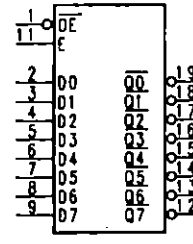
DRAWN BY: JP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: DD	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 54/74 SERIES (XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 99 OF 113



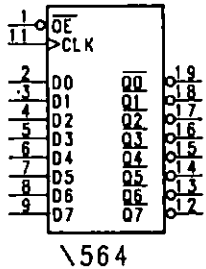
\541



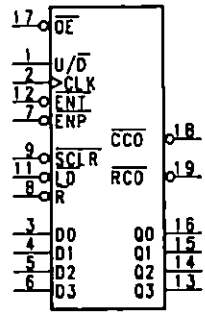
\560A



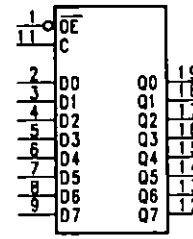
\563



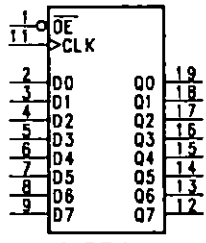
\564



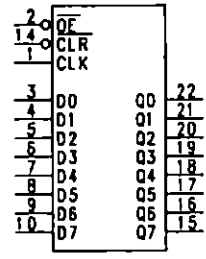
\568



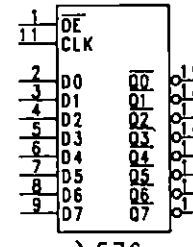
\573



\574

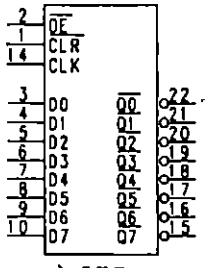


\575

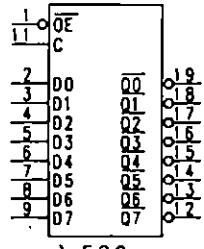


\576

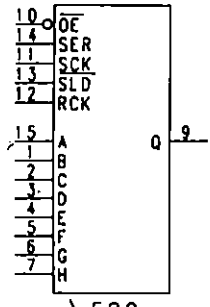
DRAWN BY: J.P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460	
CHECKED BY:	DATE 1-3-90		
APPROVED BY:	DATE 1-3-90	TITLE 54/74 SERIES (\XXX)	
REVISION:	DATE		
JOB NAME:		REV. A	SCALE 1:1	SHEET 100 OF 113



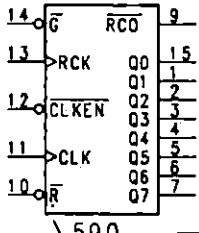
\577



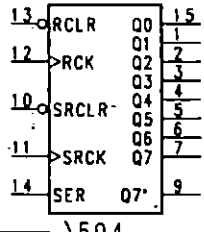
\580



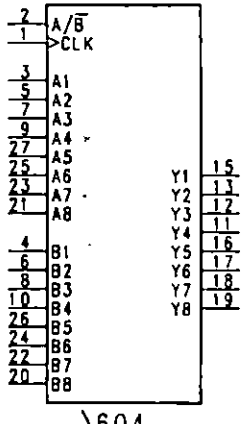
\589



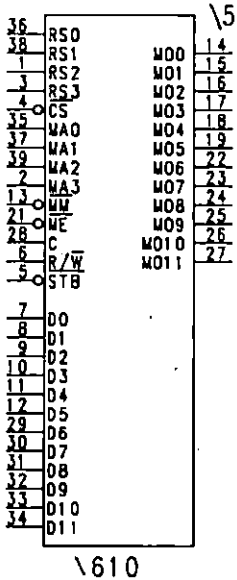
\590



\594

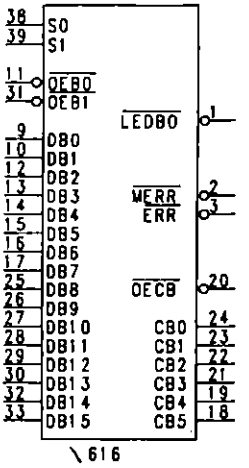


\604

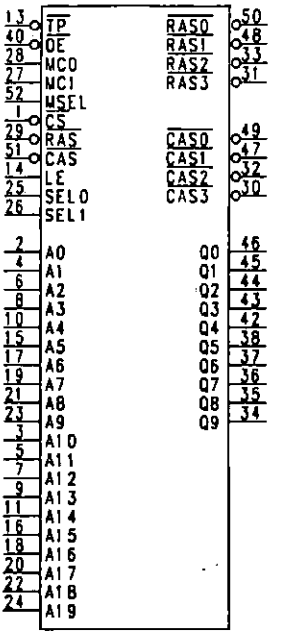


\610

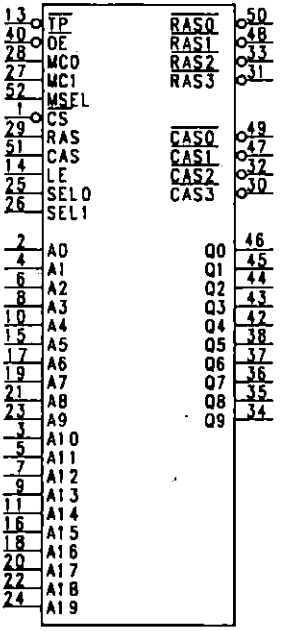
DRAWN BY: LP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460	
CHECKED BY: DC	DATE 1-3-90		
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 54/74 SERIES (\XXX)	
REVISION:	DATE		
JOB NAME:	REV. A	SCALE 1:1	SHEET 101 OF 113



\ 616

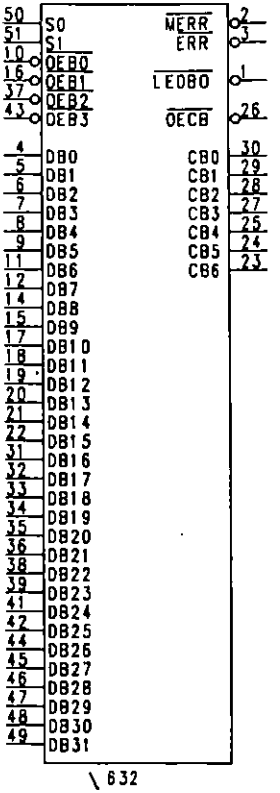


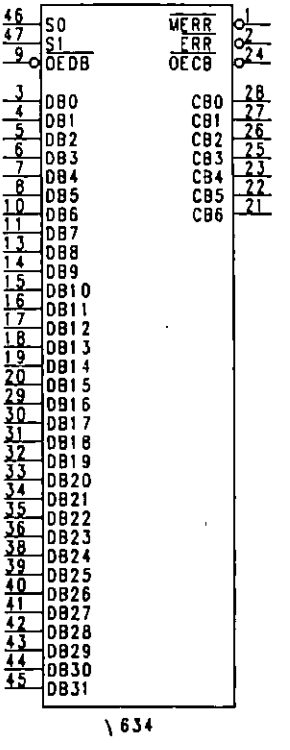
\ 6301

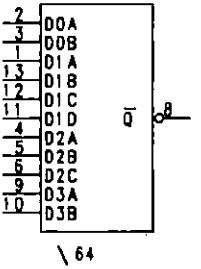


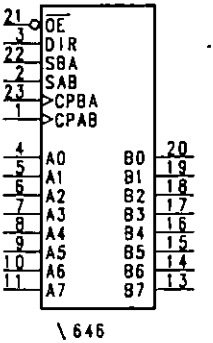
\ 6302

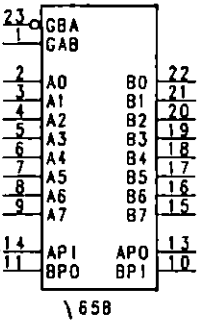
DRAWN BY: P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 64/74 SERIES (XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 102 OF 113



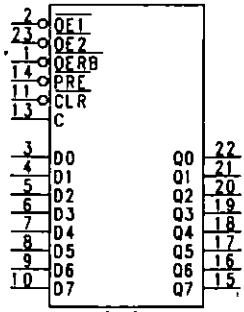




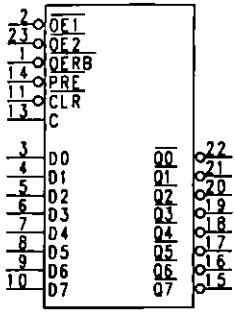




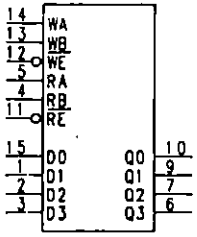
DRAWN BY: RP.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: DC	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 54/74 SERIES (\XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 103 OF 113



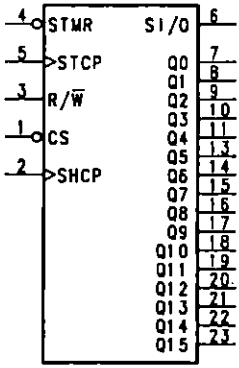
\666



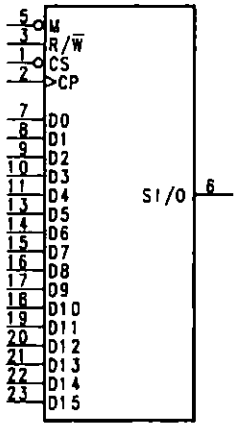
\667



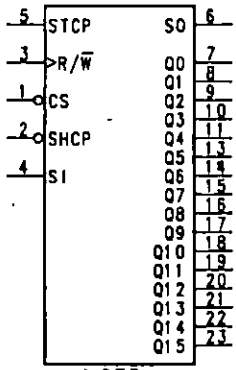
\670



\673

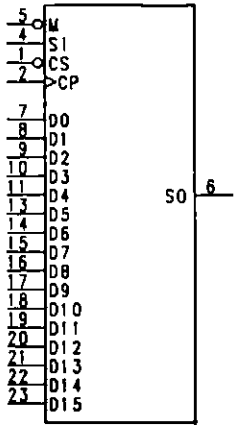


\674

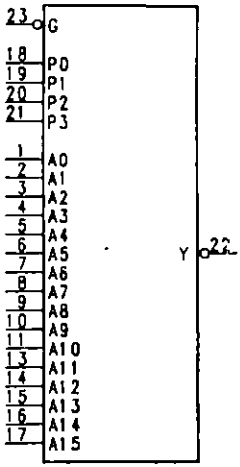


\675

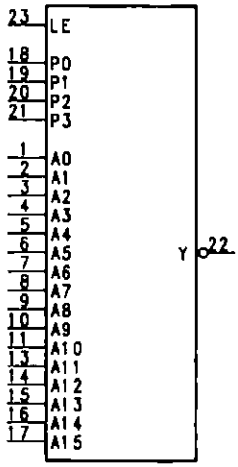
DRAWN BY: S P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 54/74 SERIES (\XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 104 OF 113



\676

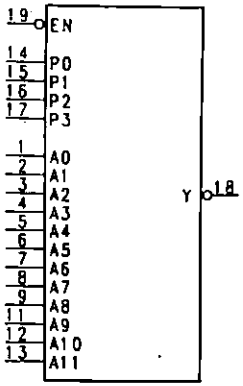


\677

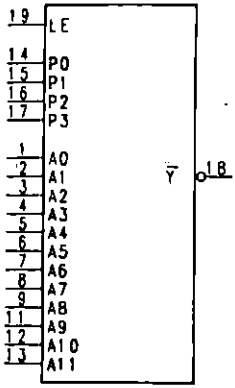


\678

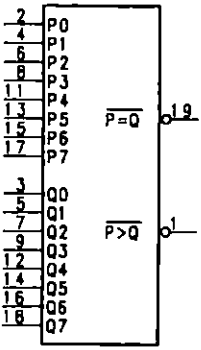
DRAWN BY: LP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: @	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 54/74 SERIES (\XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 105 OF 113



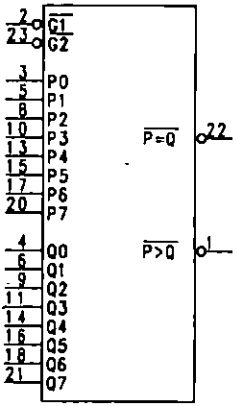
\679



\680

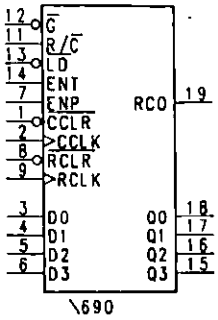


\682

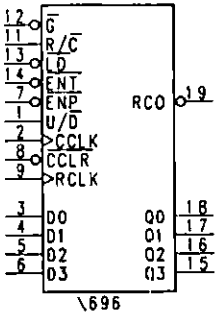


\686

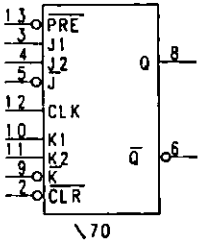
DRAWN BY: J. P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 54/74 SERIES (\XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 106 OF 113



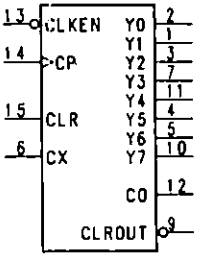
\690



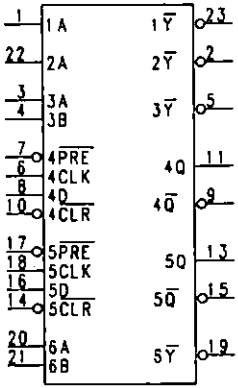
\696



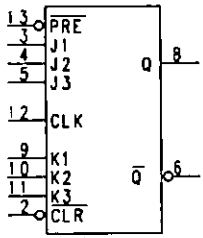
\70



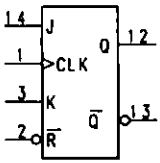
\7022



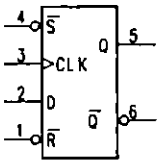
\7074



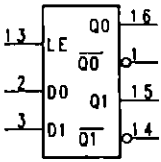
\72



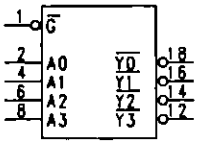
\73



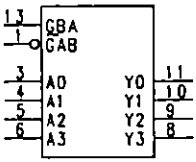
\74



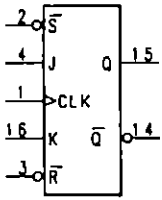
\75



\756

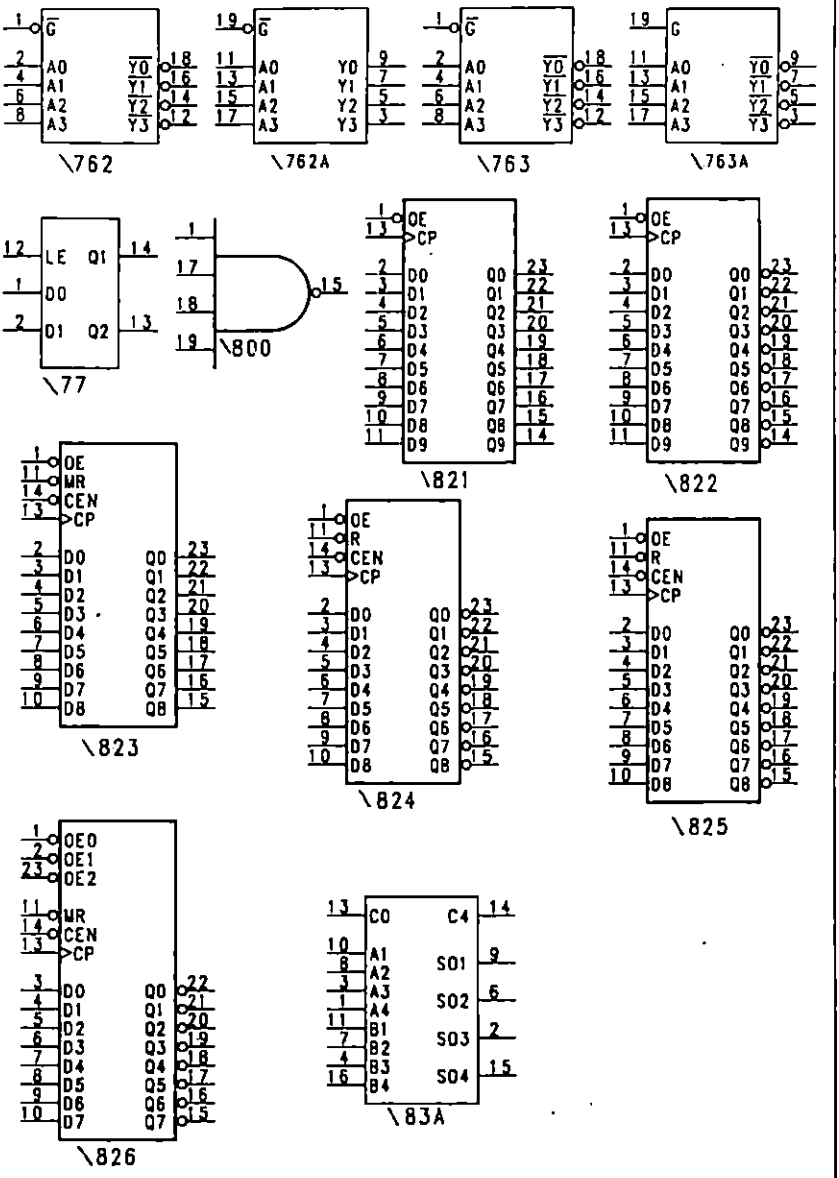


\758

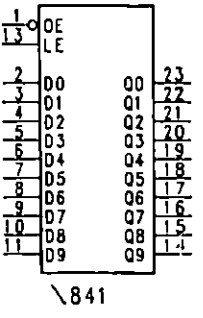


\76

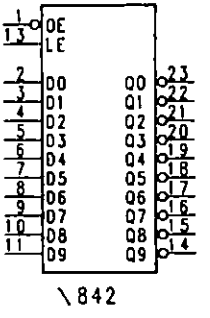
DRAWN BY: JP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: DD	DATE 1-3-90			
APPROVED BY: Gus	DATE 1-3-90	TITLE 54/74 SERIES (XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 107 OF 113



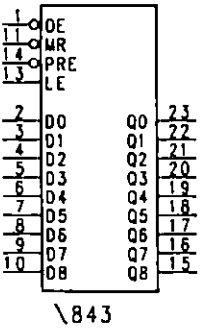
DRAWN BY: K. P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 54/74 SERIES (\XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 108 OF 113



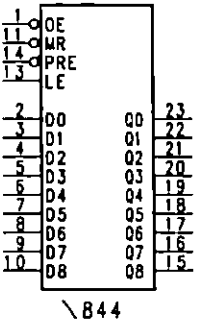
\841



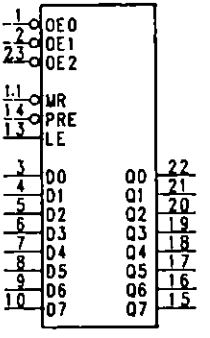
\842



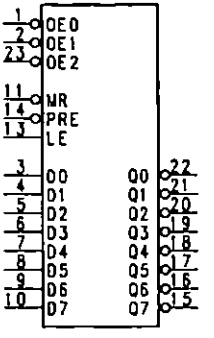
\843



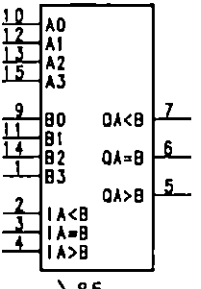
\844



\845

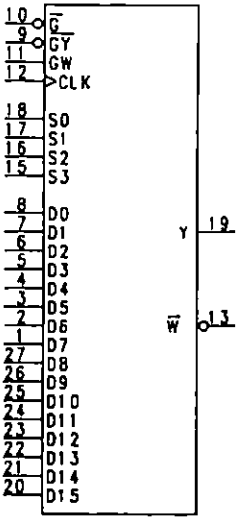


\846

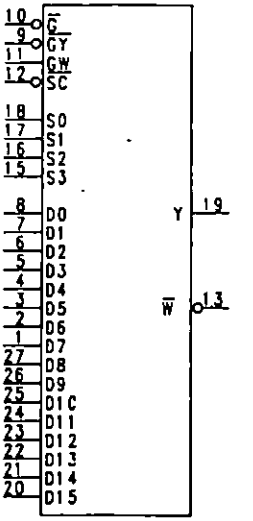


\85

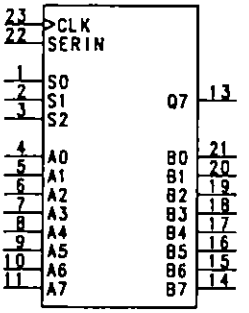
DRAWN BY: LP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: RDD	DATE 1-3-90			
APPROVED BY: RDD	DATE 1-3-90	TITLE 54/74 SERIES (\XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 109 OF 113



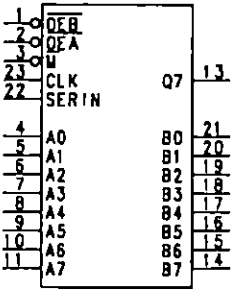
\850



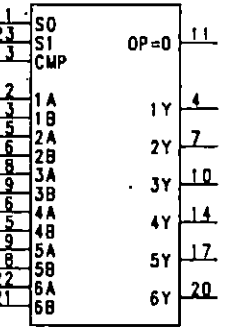
\851



\852

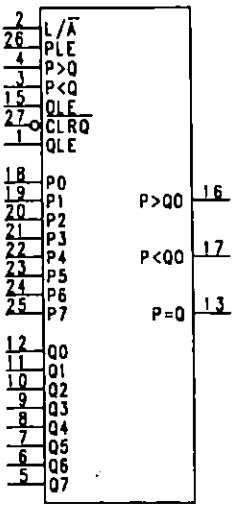


\856

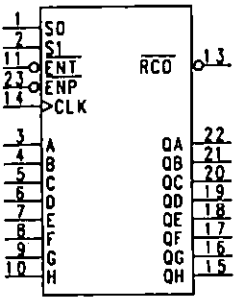


\857

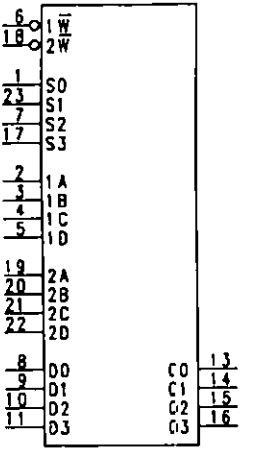
DRAWN BY: J. P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 54/74 SERIES (\XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 110 OF 113



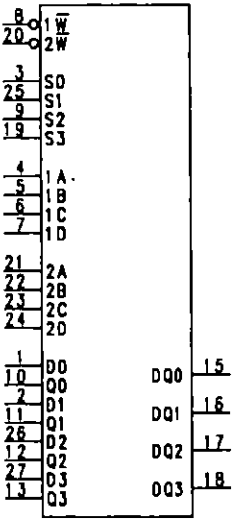
\866



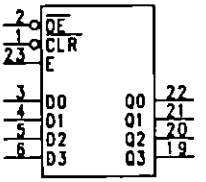
867



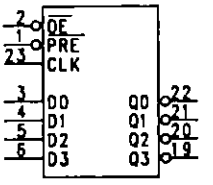
\870



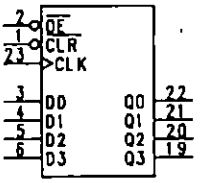
\871



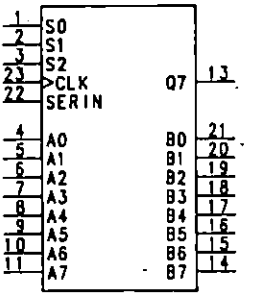
\873



\876

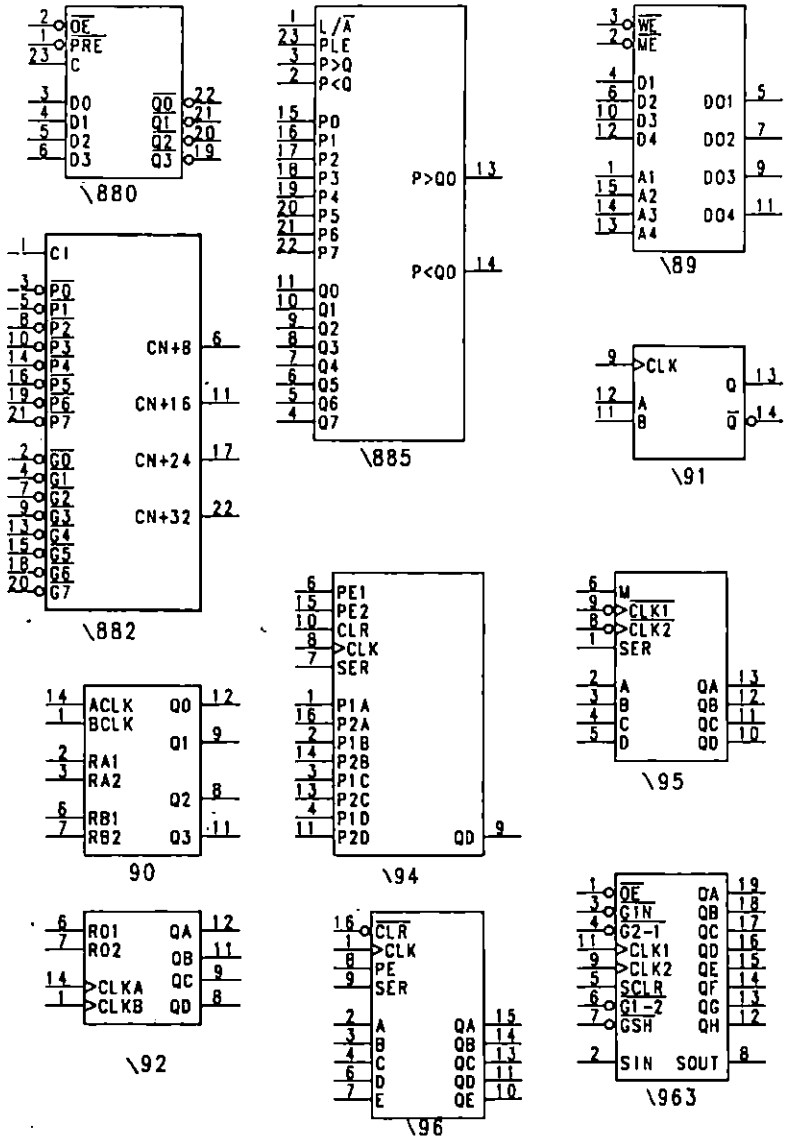


\874

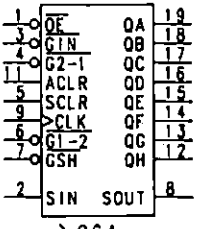


\877

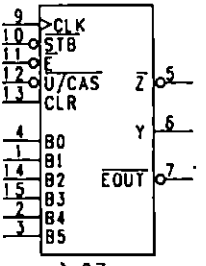
DRAWN BY: JP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE - 54/74 SERIES (\XXX)		
REVISION:	DATE			
JOB NAME:		REV.. A	SCALE 1:1	SHEET 111 OF 113



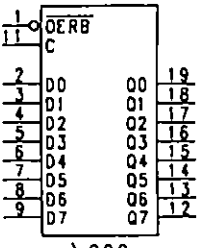
DRAWN BY: J.P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 54/74 SERIES (\XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 112 OF 113



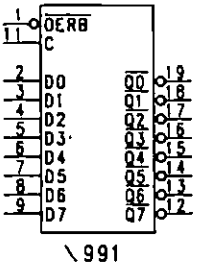
\964



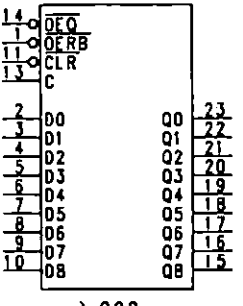
\97



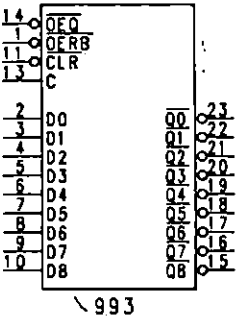
\990



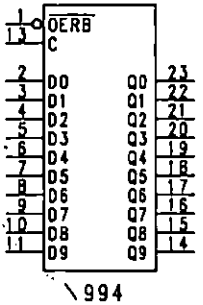
\991



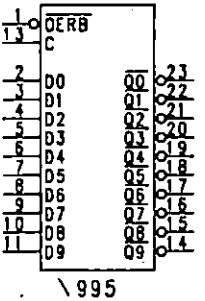
\992



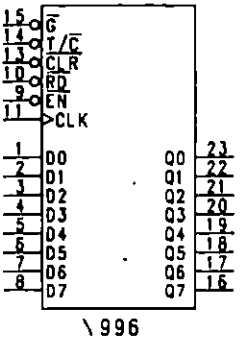
\993



\994



\995



\996

DRAWN BY: fp	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 54/74 SERIES (\XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 113 OF 113

APPENDIX A10

PADS-THERMAL

1 MARCH 1990

CAD Software, Inc. Copyright (C) 1990 -- All rights reserved.

This program is the property of and embodies the proprietary trade secrets of CAD Software, Inc. Unauthorized use and copying are prohibited by law, including United States and foreign copyright law.

PURPOSE

The purpose of this appendix is to describe the interface between PADS-2000 and the PADS-THERMAL program.

INSTALLATION

Refer to the PADS INSTALLATION PROCEDURE "PINSTALL."

DIRECTORIES

PADS\THERMAL uses the following directories:

\PADS: is always where the Thermal Analysis program resides.

\PADS\THERMAL: is where the user generated files reside (i.e., .ASC, .INP, etc.)

FILE CREATION

The PADS-THERMAL program requires a PADS-2000 ASCII format file. The creation of this file is achieved by activating the job file in PADS-2000, and selecting ASCII OUT, and selecting the ALL function. The user then proceeds to specify an output filename. This file is automatically saved to the PADS\FILES directory. It is recommended upon creation of the ASCII file, that the user specify a path to the \PADS\THERMAL directory. Upon creation of the ASCII file in PADS-2000, if the user has not specified a path to the PADS\THERMAL directory, the file must be transferred via the DOS copy command from the default PADS\FILES directory to the \PADS\THERMAL directory.

INTERFACE

Once the ASCII file has been generated, it is then run through a translator which extracts the data required for the PADS-THERMAL program. To run the translator, change the directory to \PADS, and enter PADS-2000 by typing: 2000. From the Shell, you will then, with the left button of your mouse, select "THERMAL." At this point a second menu appears. You are given the option of ITF (Interface which converts the ASCII file for the THERMAL program) or THERMAL, which is the actual program.

Upon selection of ITF, a prompt for the filename will appear; enter the name of the ASCII file.

There are three items which are extracted when the file is translated. They are:

1. "PARTTYPE* - logical part name;

2. *PART DECAL* - outermost physical dimensions of the component, and the pad locations;

3. *PART* - placement data. The part section describes the component X and Y location, rotation, the side of the board which the component resides in the design, and whether an alternate decal is being used.

TESTING THE INSTALLATION OF THE PADS-THERMAL SOFTWARE

Prior to using the PADS-THERMAL software on a design, you must assure that the software has been installed correctly, and the program is recognized by the PADS security key. To do this, we have included an ASCII file of one of the PADS-2000 training boards, as well as the files required to run the analysis. Using this file, PLACE1.ASC, you will:

1. Convert into PADS-THERMAL format
2. Load the file into the Thermal software
3. Examine the placed file
4. Run analysis

Once again, this procedure is for the specific purpose of testing the installation of the PADS-THERMAL software. This is not to be a substitute for the actual tutorial. For a comprehensive tutorial, refer to Chapter 3 of your PADS-THERMAL manual. Let's proceed:

At the DOS prompt C:\> TYPE : **2000**

Appearing on the screen will be a menu with the options shown; using your mouse, select THERMAL.

At this point a second menu appears. You are given the option to select ITF (Interface) or THERMAL.

Select: ITF

Upon selection of ITF, a prompt to enter ASCII OUT file name will appear.

TYPE: PLACE1

The following will appear at the CRT:

Converting	*PADS-2000*	data
Converting	*PCB*	data
Converting	*MATRIX*	data
Converting	*LINES*	data
Converting	*PARTDECAL*	data
Converting	*PARTTYPE*	data

```
Converting    *PART*              data
Converting    *ROUTE*             data
Converting    *END*          0    data
              *ENDFILE*
```

PREPARING PLACE1.INP FILE

CONVERTED SUCCESSFULLY

Once the conversion is complete, the menu will re-appear, you may select THERMAL.

At this point the Thermal program is visible at the CRT. There are two methods of making selections from the menu:

1. Select the first character of the command to execute the command, or;
2. Using the left and right arrow keys, move the highlight bar across the menu, and select <ENTER> to execute the command.

In this procedure, we will use method #1; proceed as follows:

SELECT: F (file)
L (load)

The following will appear at the CRT:

LOAD FROM FILE:

TYPE: **PLACE1**

LOADING FILE, PLEASE WAIT . . .

```
.INB   FILE LOADED
.INP   FILE LOADED
.INL   FILE LOADED
.GRF   FILE LOADED
```

A few moments later the following message appears:

"Component descriptions are not complete, you should go to the Board Component command immediately. Hit Return to Continue".

SELECT: <ENTER>

SELECT: B (Board)
C (Component)

The following will appear at the CRT:

Every component in .INP will be searched from .INL file.
If not found, search Main Library. When a component is taken from Main Library, you may review/set its Power Dissipation.

If you want to do it, Press Y.
Otherwise Rtn.

SELECT: Y (Yes)

"The file .INL has been fully established. Hit Return to continue".

R (Return)
R (Return)

Confirm there are parts on the board by selecting:

P (Placement)
1 (1st _ side)

Parts should be visible at the CRT. SELECT:

<ENTER>
R (Return)
R (Return)
A (Analysis)

Once the Analysis is complete, you may review its output.

O (Output)
T (Thermal - map)
B (Board - show board temperatures)
1 (1st _ side)
<ENTER>
<ENTER>

You have completed the Thermal Analysis Test procedure. You may continue to examine the menus or exit the program. Upon exiting the last menu, you must select:

Q (Quit)

You will be prompted:

Press "N" if you do not wish to save data, any other key to save, then quit. This is left up to the discretion of the user.

Please proceed with the Tutorial in the PADS-THERMAL Manual.

NAMING CONVENTIONS

It is important that the user maintain the same part type names throughout their design, i.e., schematic to board to thermal analysis. Failure to be consistent will cost time and effort editing files.

The user is advised to review the library listings in both the PADS-2000 manual, and the PADS-THERMAL manual, and decide on a common naming convention. The PADS-2000 library listing may be found in Volume 2, Reference Manual, Appendix A9. The PADS-THERMAL library listing may be found in the PADS-THERMAL manual, Appendix III.

Generic naming conventions are not recommended (i.e., 7404) for parttype names when planning to output to the Thermal program. The user should be specific with the parttype description, (i.e., 74ALS04) because Thermal Analysis must treat them differently due to their different power dissipations. In some instances, however, PADS-2000 has used a very generic approach to parttype names, specifically discrete components. This is due to the large amount of components which will use the same package, but carry a different components. For example, a PADS-2000 resistor is named R1/4W, but can have a multitude of values, such as 1K, 10k, 100k, etc. In Thermal Analysis the R1/4W,1K and R1/4W,10k are treated the same since they have the same outline, and the same nominal power dissipation of 1/4W. Therefore, it is recommended the User omit the 1K and 10k notations, but maintain the R1/4W name which resides in the library. To omit

these values from the ASCII file, it will be necessary to use an editor such as PEDIT, the CAD Software supplied editor.

NOTES:

1. Prior to selecting ASCII OUT, be sure to set the board origin to the lower left corner such that the entire board is contained in the positive, upper right Cartesian quadrant.

2. PADS-THERMAL uses the PCB DECAL origin and component outline to calculate approximate locations within PADS-THERMAL. If more than one outline exists, the first CLOSED item in the ASCII file will be interpreted as the outline.

3. PADS-THERMAL requires a component outline and component pads in order to calculate the approximate position of the component. Therefore, prior to selecting ASCII OUT, insure that all components have a part outline. Those components which might not have an outline, such as an edge connector, will require one. It is not required for the outline to be resident on layer zero or layer one. Upon conversion of the ASCII data, if more than one component outline exists, the first "closed" line item in the ASCII file of that particular component, will be used for the calculation of the component location. For additional information regarding the PADS-2000 ASCII format, refer to document number 12386, Interface Format Specification for PADS-2000.

4. Components that have been placed in the PCB design at angles other than 0, 90, 180 and 270 degrees will snap to the nearest 0, 90, 180, 270 degrees in PADS-THERMAL.

ERROR MESSAGES

"PART DETAILS NOT FOUND, REFERENCE NAME = XXX"

Problem: Part missing from the PADS-THERMAL library.
Action: Enter data into PADS-THERMAL library.

APPENDIX A11

PADS-2000 HI-RES

APPENDIX A11

PADS-2000 HIRES

1.0 HARDWARE REQUIREMENTS

The following hardware is required in order to run PADS-2000 HIRES:

PEPPER SGT-PADS graphics card with 4 megabytes of memory.

A variable scan style monitor capable of 1024 x 768 resolution.

IBM compatible VGA card*.

VGA loop-thru cable*.

*If using PADS-LOGIC and/or PADS-2000 SUPERROUTER, it will be necessary to install a VGA board (for EGA, see below) into the system, along with the SGT-PADS board. The SGT-PADS board will accept a VGA video signal and pass it directly through to the single monitor, using the loop-thru cable from Number Nine.

An EGA board may be used in conjunction with the PEPPER SGT-PADS card. This configuration requires either the use of two monitors or the swapping of monitor cables. Contact Number Nine Computer with any configuration questions.

1.1 INSTALLATION

PADS-2000 HIRES is a high performance option for PADS-2000. The software is installed by inserting Main Disk 1 in the A drive and typing A:PINSTALL. PADS-2000 HIRES installs exactly like PADS-2000.

NOTE: IT IS NECESSARY TO LOAD THE PADS-2000 PROGRAM DISK AND SET THE RESOLUTION WITHIN THE HARDWARE SETUP.

1.2 MEMORY CONFIGURATION

The SGT-PADS board has 4 megabytes of memory which allows for a drawing surface that is 4096x2048 (rectangular) in size.

1.3 /V COMMAND SWITCH

The rectangular drawing surface can be configured to be horizontal or vertical. The default is horizontal to fit an entire PC size design into the SGT-PADS' memory. Typing P2000 to enter PADS-2000 uses the default horizontal bitmap.

The drawing surface can be switched to a vertical position to display and edit vertical sections of the board design which may not completely fit in a horizontal orientation. An example of this would be a board design that is taller than it is longer. To change the orientation of the drawing surface, type: P2000/V to enter PADS-2000, where /V indicates a vertical drawing surface.

This command does not affect the display of the board; the x-axis is still horizontal, and the y-axis is vertical.

1.4 HARDWARE PAN

One of the most important features of PADS-2000 HIRES is the ability to "Pan" around the board design without having to redraw.

To view other areas of the design, move the mouse in any direction. The screen will move dynamically to the desired viewing area by following the action and direction of the mouse. The faster the mouse is moved, the faster the screen will pan.

This hardware panning can be performed at any time the gray square (screen) is smaller than the solid red rectangle (drawing surface) in the PADS-2000 viewing map area.

1.5 NEW MENU HIGHLIGHT BOARD SELECTION METHOD

When using PADS-2000 HIRES, the operator controls the cursor menu selections. There are three methods that can be used:

1. Hold down the middle button on the mouse and move the mouse across. This has the effect of moving the highlight bar across the menu area. Tap the first button to select the desired command.

2. Execute the desired command by selecting the corresponding function key to the command.

3. Rename the file CURSOR in the PADS directory. This will now allow the user to move the highlight bar across one selection each time the middle button is touched. DO NOT DELETE THIS FILE.

NOTE: IT IS RECOMMENDED THAT A THREE-BUTTON MOUSE BE USED WITH PADS-2000.

1.6 CHANGING RESOLUTIONS

To change the resolutions for PADS-2000 HIRES, you must run the hardware setup through the PINSTALL program on your PADS-2000 software disk. Changing to a lower resolution will provide lower screen resolution, but allow for a larger panning area and a higher workable scale factor on-screen. To change the resolution, select one of the following drivers under GRAPHICS DEVICE SETUP in Hardware Setup:

36. 1024 x 768 Number Nine Pepper Sgt Plus
37. 800 x 600 x 768 Number Nine Pepper Sgt Plus
38. 640 x 480 x 768 Number Nine Pepper Sgt Plus

1.7 VIEWING MAP

The viewing map has been modified to indicate the entire area of the board design stored in display memory and the view area which can be panned across the design. The map now consists of the following:

White Outline - This indicates the general board outline.

Solid Red Rectangle - This represents the area of the drawing surface that can be panned around in.

Gray Square - This represents the monitor screen and indicates where in the drawing area the user is viewing.

1.8 ARROW KEYS

The arrow keys can be used to pan around the display area. When the edge of the area stored in display memory is reached and the arrow key is hit, the drawing surface will be refreshed to display the new area indicated.

NOTE: Distance moved is the lessor of one-third the size of the window area or the distance to the edge of the bitmap, thereby preventing a repaint of the bitmap area.

1.9 HOME KEY

Pressing the HOME key redraws the PCB Layout to fill the entire size of the drawing surface, regardless of what zoom factor the screen was currently in. The display screen is also centered.

1.10 PgUp-PgDn

PgUp and PgDn work the same as in the standard PADS-2000. Using PgUp and PgDn will rescale the desired area to the drawing surface and center the display.

1.11 DEL KEY

Works the same as in standard PADS-2000. In HIRES, the display memory will be refreshed.

1.12 END KEY

Causes the screen to be repainted at the present level.

1.13 INS KEY

Centers the current cursor location on the screen. Also when in the SHOVE (F4) Routine under ROUTE (F5), the key will setup a new Route Box in that location when selected.

1.14 CTRL/HOME

By simultaneously holding down Ctrl and Home, the PCB design is toggled between a full screen view of the design and HIRES mode with panning and scrolling.

APPENDIX A12

PADS-DXF

APPENDIX A12 - PADS-DXF

TABLE OF CONTENTS

OVERVIEW 3
INSTALLATION 4
RUNNING PADX EXPORT 6
AUTOCAD HEADER VARIABLES 7
LAYERS 8
FREE TEXT 9
AUTOCAD TEXT STYLES 10
LINE ITEMS 11
DECALS 11
SPECIAL DECALS - THE VIAS 12
BODY OUTLINE 13
PADS 13
PADSTACKS 16
PARTTYPES 17
PARTS 17
TRACKS 17
AUTOCAD TO PADS (Import) 18
RUNNING PADX IMPORT 19
TRAPS FOR THE UNINITIATED 20
ERROR TRAPPING 21
MISCELLANEOUS 23

OVERVIEW

PADX is a translation program for porting data from PADS-2000 ASCII files to and from AutoCAD .DXF files.

To the PADS user, this is largely automatic. You simply select a file and run. Consequently, most of this manual covers the way the PADS database information is converted to AutoCAD format, and how to manipulate it once it is in AutoCAD.

For PADS to AutoCAD, the complete PADS board layout is converted into AutoCAD format, with as much name/type information preserved as possible to simplify post processing.

NOTE: When a PADS-2000 ASCII file is converted into the AutoCAD format, it must remain in this format. The PADS-DXF program was not designed to convert this type of file back into the PADS-2000 ASCII format. Below is a description of the AutoCAD to PADS translation program.

AutoCAD to PADS processing of TEXT, 2-D Lines and DECAL blocks is also supported, limited only by PADS database restrictions on these items. Import is only supported for V2.X or higher for PADS-2000. AutoCAD can speed creation of complex DECAL shapes and is especially good for ARC / Array style decals.

This feature can be extremely useful to create mechanical placements and outlines. This allows the design to proceed in parallel with the Mechanical Department.

INSTALLATION

What you need:

PADS-2000
AutoCAD 2.6, or higher
640K memory
Hard disk with at least 1 MEG Free
A reasonable understanding of the AutoCAD program

Other CAD programs apart from Auto CAD may be suitable, providing they accept a .DXF file input and allow custom text fonts.

PADX comes as an .EXE file and several support files (*.SHX, *.SHP).

The support files (*.SGX) are AutoCAD specific, and must be installed in the AutoCAD support file directory (i.e., where all the other *.SHX files are). AutoCAD must be able to find these files without user intervention.

NOTE: These support files (*.SHX) will be loaded automatically through the PADS-DXF installation. The program will copy the *.SHX and *.SHP files to the AutoCAD directory you specify. It will also copy the program and documentation files to the directory \PADS. If either of these directories do not exist, they will be created.

EXAMPLE: A:INSTALL \ACAD \PADS

The Autoexec.bat file will need to be modified manually to reflect the .SHX directory.

EXAMPLE: Set acad = d:\acad\shx (in the Autoexec.bat file).

Alternatively, they can be placed in the AutoCAD root directory, though that is not good practice.

You should also give AutoCAD some scratch pad memory to use while it is performing a DXFIN operation. This is best set to the maximum to avoid any "out of memory messages." It is also handy, if you are compiling AutoCAD shape files (*.SHP ---> *.SHX).

EXAMPLE: Set acadfreeram = 20 (in your autoexec.bat file)

The .SHP source files for the custom files are supplied for reference and/or modification. They have been optimized for plotting 2:1 using a 12 thou pen. If you are plotting 4:1, the first entry in each of the files must be modified slightly. See the AutoCAD manual Appendix B for details on .SHP files.

Do not be put off by AutoCAD's token warning--shape files are really very straight forward.

On the PADS-DXF program diskette, there is an EXAMPLE directory. Here is an explanation of the files:

AutoCAD to PADS files:

PADXDEMO.DWG	AutoCAD master example - Correct units
PADXDEMO.DXF	Above after DXFOUT
PADXDEMO.FXD	Above after PADX translation (PADS-2000 ASCII format)
PADXDEMO.DEC	Decals contained in PADXDEMO
PADXDEMO.JOB	Job file after ASCII in of .FXD file NOTE: Text and 2-D Lines have been added to this file.
DECALDMO.ASC	Example of decals made in AutoCAD
LOGODEMO.FXD	Example of logo in 4 different sizes made in AutoCAD

PADS to AutoCAD files:

2000A.ASC	PADS-2000 ASCII file
2000A.DXF	above after PADX translation
2000B.ASC	PADS-2000 ASCII file
2000B.DXF	above after PADX translation
PCBA.ASC	Version 2X PADS-PCB ASCII file
PCBA.DXF	above after converted through PADS-DXF
PCBB.ASC	Version 3X PADS-PCB ASCII file
PCBB.DXF	above after converted through PADS-DXF

NOTE: PCBA and PCBB files are solely used with PADS-PCB software.

These files can be studied, for help in understanding the use of the PADS-DXF program.

RUNNING PADX EXPORT

Running PADX is a straight-forward exercise. The process goes as follows:

- ASCII out your PADS-2000 job; select the ALL option from the pop-up menu.
- Output the ASCII files directly to your AutoCAD working directory.
- Exit PADS-2000.
- Run PADXINCH or PADXMETC

PADXINCH converts the coordinates to and from AutoCAD using inches/mils.

PADXMETC converts the coordinates to and from AutoCAD in metric units.

You will notice PADX creates two output files - a .DXF file and a .SCR file. The .DXF file is the main input file into AutoCAD. The script file is output only when PADX is unable to trim very short track segments. This is then used to automatically trim the tracks within AutoCAD.

To get the .DXF file into AutoCAD, you should follow the procedure below:

- Move the .DXF (and optional .SCR) files output by PADX to your AutoCAD working directory.
- Start AutoCAD.
- Select option 1 from the main menu (Create a new drawing).
- Enter your desired file name and directory.

- Use the AutoCAD DXFIN command. Respond with the .DXF file name and path (no extension necessary).
- AutoCAD can take a few minutes to read a .DXF file.
- If a SCRIPT file was generated, this should now be run using AutoCAD script command.

NOTE: If a SCRIPT file is input on top of a DXF file, the information input from the DXF file will disappear. To reactivate, go into ACAD's MODIFY LAYER and turn the items back on.

- You now have a complete PCB design in AutoCAD.

AUTOCAD HEADER VARIABLES

PADX translates the following PADS-2000 system variables:

- USERGRID - Becomes the AutoCAD SNAP setting.
- DOTGRID - Becomes the AutoCAD GRID setting.
- MAXIMUM LAYER - Used to determine the number of TRK levels and the Bottom PAD level.
- JOBTIME - Passed through for time accounting.
- REFNAMESIZE - Used for part reference names.
- $EXTMAX,$EXTMIN - Calculated by windowing the board outline and adding a bit to allow for "Reduce to" notes, etc. This allows a reasonable approximation for the initial draw. AutoCAD will always do a ZOOM EXTENTS however.
- $FILLMODE - Set OFF for faster regen; this affects Oval Fingers, Rectangular Fingers and Tracks.

LAYERS

PADX attempts to preserve as much of the PADS-2000 layer information as possible. It maintains all the user defined colors, with the exception of the background color (unless it is black). In the case of a non-black background color, any black layers take on the background color; and the background color becomes the AutoCAD installed background color (assumed to be black). PADX scans all layers except the track layers and determines which ones are in use. Layers in use, but with no color assignment are given a random color and turned OFF. Layers in use with a color assignment are turned ON.

PADX translates all PADS-2000 levels to named AutoCAD layers, except for the VIA levels. (See Special DECALS - The Vias for explanation.)

PADS-2000 Entity	Converted to AutoCAD Entity/layer.
TEXT	Simplex Text, which identified by style name; text size compensated by PADX and placed on layers TXTn.
2D LINES	Polylines of real width on layers LINn.
COPPER	Polyline outline of real width on layers CPRn.
COMBINED ITEMS	Combined 2D lines/Text convert to a block.
DECALS	Nested Blocks on Layer COM1, 2 of PADS, 2D Lines and Text.
PADS	Sub Blocks on layers PADn, block suffixes -T, -B, -I.
2D LINES/TEXT	Sub Blocks on layer 0, block suffix -D.
PARTTYPES	Not converted.
PARTS	DECAL XY Location and REF DESC (Ref Text on layer COM3, 4).

TRACKS	Routed - Polylines of real width(s) on layers TRKn with TRACK end trimmed by 15 mil from PAD Center. Unrouted - Polylines on layer TRK0 and issues warning.
VIAS	Via style on layer PADn.

It also adds a few layers of its own. These are:

1. Layer 0	- Used for block creation (A "magic" layer in AutoCAD).
2. CPR0..CPR30	- Used for copper creation.
3. LINO,TEXTO	- Used to represent PADS-2000 "On all levels" entities.
4. PAD I	- Used to represent PADS-2000 "PAD on inner levels."
5. DRL?? . . .	- Used to display drilling diagrams (V3.0 only).
6. VIA_T	- Used to represent thru vias on regular layers.
7. VIA_B	- Used to represent buried vias on regular layers.
8. VIA_M	- Used to represent micro vias on regular layers.
9. TRIM	- Used to truncate tracks whose new end points cannot readily be calculated.

IMPORTANT:

PADX Relies on the PADS-2000 MAXIMUMLAYER variable to be accurately set, as it is used to determine the number of TRK layers in use, as well as defining which PAD level is the bottom level. Please ensure this variable is in accordance with your design.

FREE TEXT

PADX places PADS-2000 text on its corresponding AutoCAD layer[TXTnn]. PADS-2000 "Text on all levels" is placed on layer

TXR0. Rotation and mirroring characteristics are preserved; text width is not (See STYLES). PADX uses the AutoCAD SIMPLEX font (RomanS in AutoCAD 9+) as a reasonable approximation of the PADS-2000 CAMFONT font. Note, however, that SIMPLEX is proportionally spaced, whereas CAMFONT is not. This will cause a small difference in the overall length.

AUTOCAD TEXT STYLES

AutoCAD does not directly support different text widths, so PADX creates AutoCAD text styles based on the PADS width used.

PADX uses the AutoCAD SIMPLEX font with style names indicating the PADS-2000 text width, e.g.:

Style name W10 indicates the PADS-2000 width was 10 thou.

These style tags allow easy substitution of a different font using the AutoCAD STYLE command, e.g.:

To change style W10 from SIMPLEX (a single stroked font) to ROMAND (a double-stroked font in Release 9+) to get a closer approximation of large TEXT to the original PADS-2000 drawing, the dialogue will go like this:

AutoCAD Command: STYLE Text style name (or ?) <current>
W10
Font file <SIMPLEX> ROMAND
Height <default> (Return)
Width <default> (Return) . . . etc.

You will also notice that the AutoCAD text has a different method of height description. PADS-2000 includes the descenders in its measurement, while AutoCAD does not. PADX automatically compensates for this during the translation in both directions.

Along with the text styles, PADX also creates a number of its own styles used for the generation of regular pad shapes (Round, square,...etc.). The style names are fairly self-explanatory, i.e.:

ROUND_DONUT - Used for drilled round pads
ROUND_SOLID - Used for undrilled round pads. . . etc.

By default, these reference outline only fonts (???-OUTL.SHX) to help with regen/redraw times. Again, the style command can be used to change these easily. We recommend staying with outline only/single stroked fonts until plot time to keep AutoCAD as fast as possible.

LINE ITEMS

PADX converts single line items to AutoCAD polylines/nested circles. Combined line items are converted to AutoCAD layer [LINnn]. PADS-2000 "Line on all levels" (e.g., Board Outline) is placed on layer LIN0. Combined text is placed on its respective TXT layer.

Copper items are placed on PADX CPRnn and are not filled. (AutoCAD doesn't support a "paint" command, as it is vector-based - the HATCH command is used instead. Also, PADS-2000 does not export the copper grid value in its ASCII file). Hatching of closed copper areas is left to the user, who can choose the hatching pattern and GRID. We recommend doing this as the last step, as it can be a slow and painful process when large copper areas are involved.

Also note that it is usually faster to hatch copper areas one at a time, rather than globally, and a lot less depressing if a "leak" is encountered.

DECALS

PADX builds a list of all decals in the *DECAL section, and then examines the *PARTTYPE and *PART sections to determine which decals are actually used. Only the decals needed are converted to blocks; the rest (usually any unreferenced alternative decals) are discarded to help minimize AutoCAD file sizes. Each decal block is composed of a number of sub-blocks to take advantage of AutoCAD nested blocks as follows:

DECAL BLOCK____ Body Outline Block

.

.

____ Terminal Block

The terminal block is composed of more sub-blocks as follows:

Terminal Block ____PadStack-1 Block _____ Pad Block-1

.

.

.

Pad Block-n

.

PadStack-y Block _____ Pad Block-n

.

.

Pad Block-n

The sections on Pads and PadStacks explain the creation of the padstack and pad blocks.

PADX examines the PADS-2000 decal name and substitutes any characters that AutoCAD considers illegal. AutoCAD will only accept numbers, letters and '$ - _' (dollar, hyphen and underscore).

PADX also tests for a 3-D tag, which can be passed into AutoCAD as height and elevation. The tag is TEXT on layer 30 starting with $.

form $ height <elevation>
e.g. decal text ' $ 500 0' on layer 30 causes PADX to create an ACAD 'cookie cutter' block with height 500 and elevation of 0.

SPECIAL DECALS - THE VIAS

PADX treats the PADS-2000 STANDARDVIA, BURIEDVIA and MICROVIA decals differently than normal decals. They normally reference sub-blocks with names S_VIA, B_VIA and M_VIA. PADX also creates another sub-block called NULL_VIA. This block exists in name only, as it contains no entities and is used to "turn off" the vias, if required. This is accomplished by redefining the STANDARDVIA, BURIEDVIA and MICROVIA to NULL-VIA with AutoCAD as follows:

AutoCAD Command: BLOCK Block name (or ?)
Standard Via=Null_Via

(Turns OFF Standard Vias)

To turn the Vias back on, redefine the blocks as follows:

AutoCAD Command: BLOCK Block name (or ?)
Standard Via=S_Via)

(Turns ON Standard Vias)

This approach negates the need for the AutoCAD equivalent of PADS-2000 layers VIA1..30. Instead, vias are created on layer VIA and placed on the TRK0..30 (depending on Track start-stop layers).

BODY OUTLINE

The body outline block is made up of polylines and/or "nested circles" to preserve the PADS-2000 line width. PADX does not use the AutoCAD DONUT option, as it tends to produce spiky circles.

The outline block has its line items on layer BDYnn when the file originates from PADS-2000. This is contrary to when a file comes from PADS-PCB V.2.31 or lower, where it would then appear on Layer COMnn.

Decal text is scaled (See TEXT description above) and placed on its respective TXTnn layer).

PADS

PADX scans all the pads in the PADS-2000 *DECAL section, identifying unique pad descriptions. All pads are converted to blocks at the lowest level of the Decal block hierarchy. The PADS-2000 pad description becomes the AutoCAD block name to allow rapid identification. This allows easy pad substitution of pads within AutoCAD and keeps the drawing database size to a minimum.

The names are made up as follows:

Shape_Size	for Round (R), Square (S) and Odd (O) pads
Shape_Size_Inner_Diameter	for Annular (A) pads (Pads 3+ only)
Shape_Size_Ori_Length_Offset	for Rectangular (RF) and Oval (OF) fingers

Examples:

R_59	Round pad, 59 thou diameter.
A_138_50	Annular ring, Outside diameter 138 thou, Inside diameter 50 thou.
RF_177_90_1000_-127	Rectangular finger, 177 thou wide, Orientation = 90 degrees, Length; = 1000 thou, Offset distance 127 thou down from drill center

Size:

The allowed size range is 0 to 500 mils. PADS with a size of 0 are not processed, while pads 1..10 are rounded up to 10.

Annular Rings Only:

- Inner Diameter:

This field was introduced in PADS-PCB 3.0 and allows the user to specify the inner diameter of an annular ring. This field is also available in PADS-2000 versions and may be quite useful when converting to AutoCAD.

Fingers Only:

- Orientation:

  This has a value of 0 or 90 degrees.

- Length:

  The total length of finger must be larger than finger width and must not exceed 1000.

- Offset:

  This is the distance between the drill center and the pad center. The range of Offset is +/- 127 or +/- half the pad length, whichever is smaller.

  For fingers with an orientation of 0 degrees, a +ve value shifts the pad right, which a -ve value shifts it left.

  For fingers with an orientation of 90 degrees, a +ve value shifts the pad up, while a -ve value shifts it down.

Construction:

- Regular Pads (i.e., Round, Square, Odd, Annular):

  Regular pads are constructed using one of several custom text fonts supplied with PADX. Calling a .SHX file improves AutoCAD regen time, minimizes database size and allows easy swapping between pads styles. Shapes also plot predictably and without fuss (unlike fat polylines). When PADX writes the .DXF file, it defaults to t???-OUTL.SHX fonts. These are a minimal representation (Outline only). You will also notice that the round pads have been replaced with octagonal pads. This has been done to speed up the AutoCAD regen time (less vectors to process) and to maximize the pad area. The pads can easily be filled in (or the shape changed, or both), using the AutoCAD style command.

AutoCAD Command: STYLE Text style name (or ?) <current>

Responding with a ? will list the styles defined. If you wish to fill the round pads and change them from octagonal to round the dialogue will go like this:

AutoCAD Command: STYLE Text style name (or ?) <current>
ROUND_DONUT
Font file <OCT-OUTL>RND.DONT

Height <default> (Return)
Width <default> (Return) . . . etc.

Accept all the remaining defaults. When you have finished AutoCAD will regen and hey presto! All the round pads are now round and filled.

- Fingers: (i.e., Oval and Rectangular Fingers)

  Fingers are made up of a number of AutoCAD solids (8 for Oval, 4 for Rectangular). They will have a hole in the center if the padstack containing the finger is drilled. As with fat polylines, solids are a bit fiddly to plot properly; and a little experimenting with AutoCAD plot parameters will pay dividends. You may also like to have a go writing a shape file and doing a block substitution if you consistently use the same format finger. (See Appendix B of the AutoCAD manual for details on how to do this). As with tracks, fingers come into AutoCAD unfilled. Using the AutoCAD FILL command will fix that (and slow down regen/redraw time). The recommendation is to leave FILL Off, until plot time.

PADSTACKS

PadStacks are converted to blocks on a global basis, and are named as follows:

PdStk-number

Example:

DIP14_PS1 is created on AutoCAD layer 0 composed of sub-blocks

S_60	on layer PAD1 (from Square PAD on PADS top level)
A_80_60	on layer PAD2..ml (from Annular ring on PADS internal levels)
R_69	on layer PADml (from Round pad on PADS levels)
R_70	on layer PAD27 (from round pad on level 27)
	(ml = max layers form ASCII header file)

Note that the padstack number does not correspond directly to the PADS-2000 padstack, as PADX identifies only the unique padstacks

within a decal and not exception cases as does PADS-2000. This means a decal such as a 120 pin Quad Flat Pack, which in PADS would have a minimum of 61 padstacks, will have only 2 in AutoCAD.

PARTTYPES

PADX only uses the PADS-2000 *PARTTYPE section to reference alternative decals. Gate swap, Pin swap information, etc., is ignored. Note, however, that the ASCII file MUST contain the *PARTTYPE section in order to process the decals.

PARTS

PADX scans the *PART section in conjunction with the *PARTTYPE section to determine which decals need to be processed. Only decals actually used in the PADS-2000 drawing are converted to blocks.

TRACKS

PADX converts PADS-2000 routed tracks to AutoCAD polylines, preserving the PADS-2000 width and level information. Unrouted tracks and the unrouted portions of partially routed tracks are placed on layer CON.

PADX truncates all tracks by 15 mils to avoid the tracks blocking the holes in donut pads. This is a major feature in PADX - for single-sided boards having pads with a hole in them greatly facilitates the fabrication of prototypes in-house.

NOTE: This feature is controlled by an entry in the PADX.CFG file called TRIMTRK and can be set to either Yes or No.

Any truncated endpoints that cannot be readily calculated (i.e., tracks that end in a tangle of short segments) are left to AutoCAD to tidy up with the TRIM command via a script file.

Note that the tracks will have sharp corners due to the way AutoCAD mitres the corners of POLYLINES. This is not usually a problem but it

pays to check the layout before plotting. Areas to be watched are closely spaced tracks of greatly different widths.

There is a fillet command in AutoCAD if this poses a problem.

AUTOCAD TO PADS (Import)

Information can also be imported FROM AutoCAD to PADS. This is used for mechanical placements and outlines, and can also be used for complex DECAL creation.

PADX Converts the AutoCAD .DXF file to .FXD (line items, etc.), and/or .DEC (Decal blocks ONLY) files. The .FXD file is exactly the same format as PADS .ASC files, the different name is used to remind users where it was derived from, and to avoid overwriting .ASC files.

The .FXD file is converted as follows:

PADX tests ACAD layer for valid number 0..30, then tries TXT0..TXT30; and if it finds neither, then the item is placed on the default layer (30).

An example is in PADXDEMO.DWG.

AutoCAD Entity Converted to PADS-2000 Entity:

TEXT	Default layer TXT30, width default = 5. ACAD has not width attribute, but PADX looks for Style Names W1..W50 convert to widths in PADS. Text height of 100 in ACAD = 143 in PADS because PADS height includes descenders, while ACAD's does not. PADX scales text at conversion to match physical size.
LINES	Default layer LIN30, width default = 10. Blocks containing only LINES/TEXT are converted to PADS-2000 COMBINED 2-D line items. 2-D lines, circles and ARCS are converted.

POLYLINES — As above only the WIDTH assignment is passed into PADS. Legal widths are 1..250 mil.

DECAL BLOCK — PADX is supplied with an AutoCAD shape file PADX-PIN that may be used to tag terminals for PADX. Shape names are T1..T250 and NAME-TYPE, and PADX looks for these and converts to the associated PADS entity. A DECAL BLOCK is named, and comprises one or more PADX-PIN shapes and 2-D lines and/or Text. PADX converts all such blocks to file .DEC, which can then be edited (e.g., change PAD-STACK info, remove comments) and the Decal(s) read directly into PLIB. Text, 2-D lines, circles and ARCS within the block convert to associated PADS-2000 entities. Width default = 10. PADSTACKS default can be changed; using PEDIT or PLIB.

RUNNING PADX IMPORT

For DECALS a typical process generally goes as follows:

- Create a drawing in AutoCAD - Use lines/polylines, TEXT to draw ARCS, circles and polygons. Locate TERMINALS using SHAPES (T1..250) from supplied PADX-PIN shape file. These SHAPEs are picked up by PADX to indicate 'this block is a decal' and where the PADSTACKS are. PADSTACKS default to 60 mil RND.

  When complete, combine the items for that decal into a block, with the required decal name. It is this block description that is converted by PADX. Do not go overboard with LINES or TEXT.

  NOTE: In previous versions of PADS-DXF, the customer had to upscale and downscale the information in ACAD. This feature is now controlled when running the PADXINCH batch file.

- DXFOUT only the sections required for PADS to avoid overloading PADS with unexpected or unscaled information (e.g., title blocks, logos, etc.). See also AutoCAD WBLOCK command.

- Run PADX.

- Convert the .DXF file using PADS-DXF.
- Load the .DEC file into PEDIT (or similar). Modify PADSTACKS, layers, line widths as required.
- After changes, save as .D file in \pads\lib.
- Start PLIB.
- Select USER, DECAL, READ option.
- Read .D file into the User Decal Library.
- Select USER, PART option and add a PART reference to the new DECAL.
- Start PADS.
- Select ON-THE-FLY, ADD PART to load new PART into PADS and check all is as expected.
- You now have a complete PART & DECAL in PADS that was made in AutoCAD.
- Repeat this for as many decals as required. More than one decal per drawing is fine, but keep the first one simple to speed learning.
- Non-decals/mechanical details are as above, without the terminals. Any block items in AutoCAD convert to PADS combined 2-D Lines and Text.

NOTE: PADS has no "uncombine" command so use blocks with care.

TRAPS FOR THE UNINITIATED

Before you start the AutoCAD mechanical drawing/decal to pass into PADS:

- The PADX disk comes with a PADXDEMO.DWG file that uses techniques in AutoCAD to assist with speed and precision. Study this first!

- Use POLYLINES in preference to lines, as these are closer to the PADS 2-D lines. Their WIDTH is passed, and the vertices are shared, which avoids 1 mil rounding errors "splitting" a drawing.

- PADS data area for ASCII input is 0..32000 (positive) so AutoCAD drawing should be in this area. -ve coordinates can pose problems.

- When doing DXFOUT, use the default 6 decimal places of precision. Do NOT select 0, as the .DXF file has Angles and Tangents in floating point format needed by PADX.

- PADX database for TEXT and line items is limited, so use simplified construction where possible--this is especially true for LOGOS. Check the PADX report screen, and the ASCII output file (.FXD) for entity size.

- Watch out for non-displayed blocks. These will be translated by PADX. See the AutoCAD WBLOCK command.

- DXFOUT ONLY those items you expressly require in PADS. This is a safe way to ensure no unwanted information is translated.

- When major PADS database limits are exceeded, PADX stops translating.

- PADS-2000 Decals, once they are converted from AutoCAD to PADS-2000, have their origins set to 0 by the Conversion routine. You should modify the first line of the Decal file, the second and third field to 1000 by 1000 so the Part will appear in the middle of the graphics editor in PADS-2000.

ERROR TRAPPING

PADX does some test while processing files and reports and errors to a file PADX.ERR.

These errors are usually WARNINGS, issued when PADX encounters a problem in the file.

Examples on Export (>ASC to .DXF):

- Unexpected end of file
  Check ASCII file for illegal characters

- Unrouted/partially routed tracks
  These are placed on layer CON

- Referenced decal not found
  The ASCII output MUST have DECALS if PARTS are output.

- nnn - Can't be converted to integer
  Check ASCII file for illegal characters.

Examples of Import (.DXF to >DEC, >FXD):

- Text is not left justified
  Text origin will change as PADS can only left justify text.
  Draw all text in AutoCAD left justified for correct placement.

- Text too large
  PADX will pass TEXT, but at maximum PADS height.

- Invalid Line/Arc/Circle - Zero length
  Check AutoCAD units. PADX round and does NOT convert zero items.

- Block nnn is hatching and will not be unprocessed
  PADX ignores hatching - do not use.

- Block nnn has no valid entities
  PADX ignores blocks with no PADS entities.

- Duplicate pin #nn in file
  PADX-PIN shapes #nn use twice or more. PADX will Auto-increment terminal numbers. (i.e., PADX-PIN tagged terminals are not discarded.

- Line limits are reached.
  AutoCAD can easily overload PADS lines database.
  This PADX check is for a guide only, and is not 100%.

MISCELLANEOUS

The .DXF file can become fairly large, generally 3..4 times the size of the .ASC file. This can pose a problem when archiving and/or transferring the file to an AutoCAD workstation.

There are a number of solutions to this:

1. Use a PACK program to squash the data before copying to floppy disk. The .DXF file uses a very loose ASCII format that will pack 86% or to 1/6 of the original size.

2. It can be cut in two at the the start of the ENTITIES section, (i.e., right after the ENDSEC of the BLOCKS section). If you take this approach, remember to add a new 0, EOF at the cutoff point. You will need a word processor in non-document mode to do this. The Entities section can be further hacked providing the appropriate File Separator Groups are inserted. (See the AutoCAD manual Appendix C for the DXF file definition). The .DXF file containing Layer and Block information must be the first file inputted. AutoCAD will ignore any subsequent block definitions.

3. ASCII Out everything but the tracks, convert to a .DXF file and then repeat with only the tracks. Again, the block/decal definitions must be DXFINed to AutoCAD first. As tracks generally make up 60..70% of the database, this is usually the best solution.

4. There is a file called PADX.CFG. This file contains the commands which control trimming the tracks and sizes of databases which can be converted.